Carlson TakeOff R1

Carlson Software Inc.

User's manual

April 28, 2006

Contents

Chapter 1. Tutorial **1**

CAD File TakeOff 2

Road Work 16

Trench Network 28

Drillhole and Strata 41

Digitizing 47

Chapter 2. AutoCAD Overview **67**

Issuing Commands 68

Selection of Items 69

General Commands 70

Properties and Layers 71

Properties 72

Chapter 3. File Menu **73**

New 74

Open 75

Close 75

Save 75

Save As 75

Plot 76

Recover 80

Audit 81

Purge 81

Store Project Archive 82

Extract Project Archive 82

Import Xref to Current Drawing 83

Xref Manager 83

Import-Export 85

Data Collectors 85

Convert LDD-AEC Contours 108

Import/Export LandXML Files 109

Import/Export Carlson Triangulation Files . . . 110
Import/Export DXF Files . . . 111
Import Polyline File . . . 111
Export Polyline File . . . 112
Export Surface DXF Files . . . 113
Export Topcon Tin File . . . 113
Clipboard . . . 116
Display Last Report . . . 118
Exit . . . 118

Chapter 4. Tools Menu 119

Drawing Cleanup . . . 120
Define Layer Target/Material/Subgrade . . . 123
Edit Selected Layer . . . 126
Set Layer For Existing . . . 126
Set Layer For Design . . . 127
Set Layer For Other . . . 127
Boundary Polyline . . . 127
Areas Of Interest . . . 128
Topsoil Removal and Replament Options . . . 130
Make Existing Ground Surface . . . 134
Make Design Surface . . . 136
Adjust Design Surface . . . 137
Overexcavate Surface . . . 137
Make Overexcavation Surface From Exsting/Design Surface . . . 137
Make Overexcavation Surface From Strata Surface . . . 138
Make Overexcavate Surface From Screen Entities . . . 138
Adjust Overexcavate Surface . . . 139
View Overexcavate Surface . . . 139
Draw Overexcavate Surface 3D Faces . . . 139
Erase Overexcavate Surface 3D Faces . . . 140
Draw Overexcavate Cut Color Map . . . 140
Erase Overexcavate Cut Color Map . . . 140
Clear Overexcavate Surface . . . 140
Surface Manager . . . 141
Make User-Defined Surface . . . 142

Set Active Surfaces . . . 142
Design Surface Vertical Offset . . . 143
Existing Surface Vertical Offset . . . 143
Merge Existing With Design . . . 143
Triangulate & Contour . . . 144
Triangulation File Utilities . . . 153
Volumes By Triangulation . . . 157
Draw 3DPoly Perimeter . . . 158
Draw 3DPoly Base Breakline . . . 159
Calculate Stockpile Volume . . . 159
Calculate Pond/Pit Volume . . . 161
Calculate Total Volumes . . . 162
Calculate Volumes Inside Perimeter . . . 168
Material Quantities . . . 168

Chapter 5. Edit Menu **177**

Undo . . . 178
Redo . . . 178
Erase, Select . . . 178
Erase by Layer . . . 178
Erase by Closed Polyline . . . 179
Erase Outside . . . 180
Move . . . 180
Standard Copy . . . 180
Copy To Layer . . . 180
Standard Explode . . . 181
Block Explode . . . 182
Align . . . 182
2D Align . . . 183
Trim . . . 183
Scale . . . 184
Extend To Edge . . . 184
Extend by Distance . . . 185
Break by Closed Polyline . . . 186
Break At Selected Point . . . 187
Break Polyline at Specified Distances . . . 187

Break at Intersection . . . 187
Change Properties . . . 188
Rotate by Bearing . . . 188
Standard Rotate . . . 189
Edit Text . . . 189
Find_and_Replace_Text . . . 189
Text EnlargeReduce . . . 190
Text Explode To Polylines . . . 190
Image Frame . . . 191
Image Clip . . . 191
Image Adjust . . . 192
Remove Groups . . . 193
Join Nearest . . . 193
Offset Polyline . . . 194
Perimeter Polylines Properties . . . 195
3D Entity to 2D . . . 195
Perimeter Polylines Properties . . . 196
Entities to Polylines . . . 196
Reverse Polyline . . . 197
Reduce Polyline Vertices . . . 197
Densify Polyline Vertices . . . 198
Draw Polyline Blips . . . 198
Set Polyline Origin . . . 199
Add Intersection Points . . . 199
Add Polyline Vertex . . . 199
Edit Polyline Vertex . . . 200
Edit Polyline Section . . . 200
Remove Duplicate Polylines . . . 202
Remove Polyline Arcs . . . 202
Remove Polyline Segment . . . 202
Remove Polyline Vertex . . . 203
Break 3D Polyline by Surface . . . 203
Tag Hard Breakline Polylines . . . 203
Untag Hard Breakline Polylines . . . 204
Smooth Polyline . . . 204
Change Polyline Width . . . 205

Check Elevation Range . . . 205
Close Polylines . . . 206
Open Polylines . . . 206
Highlight Crossing Plines . . . 206
Select by Filter . . . 208
Select by Elevation . . . 208
Select by Area . . . 209

Chapter 6. View Menu 211

Redraw . . . 212
Regenerate . . . 212
Zoom - Window . . . 212
Zoom - Dynamic . . . 212
Zoom - Previous . . . 212
Zoom - Center . . . 213
Zoom - Extents . . . 213
Zoom IN . . . 213
Zoom OUT . . . 213
Zoom Point(s) . . . 214
Pan . . . 214
Twist Screen Standard . . . 214
Twist Screen Line . . . 215
Twist Screen Surveyor . . . 215
Restore Due North . . . 215
Display Order . . . 215
Update Colors For Set Elevations . . . 216
3D Drive Simutation . . . 216
Existing Surface 3D Viewer . . . 218
Design Surface 3D Viewer . . . 219
FlyOver Along 3D Polyline . . . 221
3D Viewer Window . . . 223
Viewpoint 3D . . . 225
Layer Control . . . 226
Change Layer . . . 228
Freeze Layer . . . 229
Thaw Layer . . . 229

Isolate Layer . . . 229
Restore Layer . . . 229
Set Layer . . . 229

Chapter 7. Draw Menu 231

Line . . . 232
2D Polyline . . . 232
3D Polyline . . . 234
Circle . . . 235
Insert Symbols . . . 235
Insert Drawing . . . 237
Write Block . . . 238
Text . . . 240
Hatch . . . 241
2 Tangents Radius . . . 244
2 Tangents Arc Length . . . 245
2 Tangents Chord Length . . . 245
3 Point Curve . . . 245
PC PT Radius Point . . . 246
PC Radius Chord . . . 246
Raster Image . . . 247
Place Image by World File . . . 249
Closed Polyline By Interior Point . . . 249
ShrinkWrap Entities . . . 250
Building Envelope Polyline . . . 251
Design Pad Template . . . 252
Title Block . . . 260
Distance with Leader . . . 262
Curve_Arrow . . . 262
Bar Scale . . . 263
North Arrow . . . 263
Contour Elevation Label . . . 264
Color Contours by Elevation . . . 266
Color Contours by Interval . . . 267
Draw Triangular Surface . . . 268
Run Off Tracking . . . 269

Slope_At_Point . . . 270
Cut Fill Map Legend . . . 271
Cut_Fill_Labels . . . 271
Cut Fill Centroids . . . 273
Point Defaults . . . 275
Draw-Locate Points . . . 277
Field to Finish . . . 280
Spreadsheet Edit Points . . . 296
On-Screen Edit Points . . . 297
Scale_Point_Attributes . . . 297
Resize_Point_Attributes . . . 298
Twist Point Attributes . . . 298
Erase Points . . . 299
Create Points from Entities . . . 300
Coordinate File Utilities . . . 301
Point Group Manager . . . 308
Interpolate Points . . . 316
Spot_Elevation . . . 318
Create Points On Surface . . . 319
Set Point Elevations To Surface . . . 320
Import Text/ASCII File . . . 321
Import TDS RD5 File . . . 323
Import LDD MDB File . . . 323
Export Text/ASCII File . . . 323
Convert Points to AECC POINTs . . . 325
Convert Points to Softdesk . . . 325
Convert Points to C&G . . . 325
Convert Wild/Leica to Points . . . 326
Convert Points to Eagle Point . . . 326

Chapter 8. Digitize Menu 327

Digitizer Setup . . . 328
Tablet On . . . 329
Tablet Off . . . 329
Tablet Calibrate . . . 330
Save Tablet Calibration . . . 332

Load Tablet Calibration . . . 332
Digitizer Settings . . . 332
Existing . . . 334
Design . . . 334
Other . . . 334
Point . . . 334
Spot Elevation . . . 335
2D Polyline . . . 337
3D Polyline . . . 338
Perimeter . . . 339
Areas . . . 340
Contour Polyline . . . 341
Sections . . . 342
End Areas . . . 345

Chapter 9. Elevate Menu 347

Change Elevations . . . 348
Set Polyline to Elevation . . . 348
Edit-Assign Polyline to Elevations . . . 349
Edit Polyline Vertex . . . 350
2D to 3D Polyline by Points . . . 351
2D to 3D Polyline-By Text . . . 352
2D to 3D-By Text With Leader . . . 353
2D to 3D Polyline by Start/End Elevations . . . 355
Pad Polyline By Interior Text . . . 356
Drape 3D Polyline On Surface . . . 358
Convert Spot Elev To Points . . . 359
Assign Contour Elevations - Multiple in Series . . . 362
Assign Contour Elevations - From Contour Labels . . . 364
Assign Contour Elevations - Single Elevation Group . . . 365
Edit Contours . . . 365

Chapter 10. Inquiry Menu 367

List . . . 368
Point ID . . . 369
Layer ID . . . 369

Curve Info . . . 370
Polyline Info . . . 370
Surface Inspector . . . 370
Surface Report . . . 371
Drawing Inspector . . . 372
Layer Inspector . . . 374
Layer Report . . . 374
Hatch Subgrade Areas . . . 375
Erase Subgrade Hatches . . . 377
Hatch_Subgrade_Areas . . . 377
Label Subgrade Areas . . . 378
Erase Subgrade Labels . . . 378
List Elevation . . . 378
Distance Report . . . 379
Bearing and Distance . . . 379
Label Angle . . . 379
Polyline Report . . . 380
Slope Report . . . 382
Slope Zone Analysis . . . 383
Elevation Zone Analysis . . . 387
Story Stake From Surface Entities . . . 390
Story Stake By Points/Polyline . . . 391
Area Defaults . . . 392
Area by Inverse . . . 394
Area by Lines & Arcs . . . 394
Area by Interior Point . . . 395
Area by Closed Polylines . . . 395
Area Descriptions . . . 396

Chapter 11. Settings Menu 399

Drawing Setup . . . 400
Preferences . . . 401
Configure . . . 402
Edit Symbol Library . . . 407
Toolbars . . . 408
Text Style . . . 409

Units Control 410
Object Snap 412
Mouse Clicking Settings 415
Crosshairs 415
Set UCS to World 416
Set Environment Variables 416

Chapter 12. Drillhole Menu 419

Drillhole Strata Settings 420
Drillhole Import 422
Place Drillhole 424
Edit Drillhole 426
Reports 428
Make Strata Surface 430
Clear Strata Surface 430
Draw Strata Cut Depth Contours 431
Erase Strata Cut Depth Contours 431
Draw Strata Cut Color Map 432
Erase Strata Cut Color Map 432
Draw Strata Surface 433
Erase Strata Surface 433

Chapter 13. Trench Menu 435

Input Trench From Polyline 436
Create Trench Network Structure 437
Edit Trench Network Structure 439
Remove Trench Network Structure 440
Find Trench Network Structure 441
Export Trench Network Data 441
Trench Network File Backup 442
Draw Trench Network Plan View 442
Draw Trench Network Profile 442
Plain View Label Settings 446
Input Edit Trench Template 448
Draw Typical Trench Template 450
Trench Subgrade Areas 451

Trench Network Quantities . . . 451
Report Trench Network . . . 453

Chapter 14. Roads Menu 455

Input-Edit Centerline File . . . 456
Polyline to Centerline File . . . 462
Draw Centerline File . . . 462
Centerline Report . . . 463
Import Centerline . . . 464
Station Polyline/Centerline . . . 464
Label Station-Offset . . . 470
Offset Point Entry . . . 473
Calculate Offsets . . . 475
Quick Profile From Surfaces . . . 477
Profile From Existing Surface . . . 479
Profile from Design Surface . . . 479
Design Road Profile . . . 480
Design Sewer/Pipe Profile . . . 482
Quick Profile from Screen Entities . . . 489
Profile from Screen Entities . . . 491
Profile from Grid or TIN File . . . 492
Profile from 2D Polyline . . . 492
Profile from 3D Polyline . . . 493
Profile from Points on Centerline . . . 493
Import Profile . . . 494
Profile To 3D Polyline . . . 494
Profile To Points . . . 495
Input-Edit Profile File . . . 497
Draw Profile . . . 499
Polyline Slope Report . . . 507
Pipe Depth Summary . . . 509
Profile Report . . . 511
Quick Section . . . 512
Input-Edit Section Alignment . . . 515
Sections From Existing Surface . . . 517
Sections From Design Surface . . . 517

Sections from Screen Entities 518
Sections from Grid or TIN File 520
Sections from Polylines 520
Sections from Points 522
Import Sections 523
Sections to 3D Polylines 523
Sections to Points 524
Slope Zone Section Analysis 525
Highway Section Staging 526
Input-Edit Section File 527
Draw Section File 532
Section Report 543
Calculate Sections Volume 545
Mass Haul Analysis 546
Calculate End Area 548
Input Edit End Area File 549
Print Earthwork File Report 550
Design Template 551
Draw Typical Template 562
Template Transition 564
Input-Edit Super Elevation 567
Input-Edit Template Series 569
Topsoil Removal/Replacement 570
Assign Template Point Profile 572
Assign Template Point Centerline 573
Process Road Design 577

Chapter 15. Display Menu **587**

Existing Drawing 588
Existing Contours 588
Existing Surface 589
Design Drawing 590
Design Contours 590
Design Surface 591
Cut Fill Contours 592
Cut Fill Labels 592

Cut Fill Color Map 593
Other Drawing 594
Display Options 595

Chapter 16. Window Menu **603**

Chapter 17. Help Menu **605**

Project Checklist 606
OnLine Help 606
Training Movies 606
Carlson WebSite 606
About Carlson TakeOff 607

Tutorial 1

CAD File TakeOff

Note: Completing these tutorials will alter the drawing files (demo1.dwg, demo2.dwg, demo3.dwg). To run through the tutorials a second time, copy over the original drawing files (.dwg) provided on the CD under "Tutorials". In addition, all demo .flt, .tin, .trg, .ini, .cl, .pro, .lot, .tpl, .bak, .sew, and .tch files need to be deleted from the folder under C:\Program Files\Carlson TakeOff R1\WORK. If you have your own drawing files, be sure to only delete files named demo1, demo2, or demo3, and not a file attached to one of your own drawing.

This lesson takes a drawing file from cleanup to volume calculations and surface viewing.

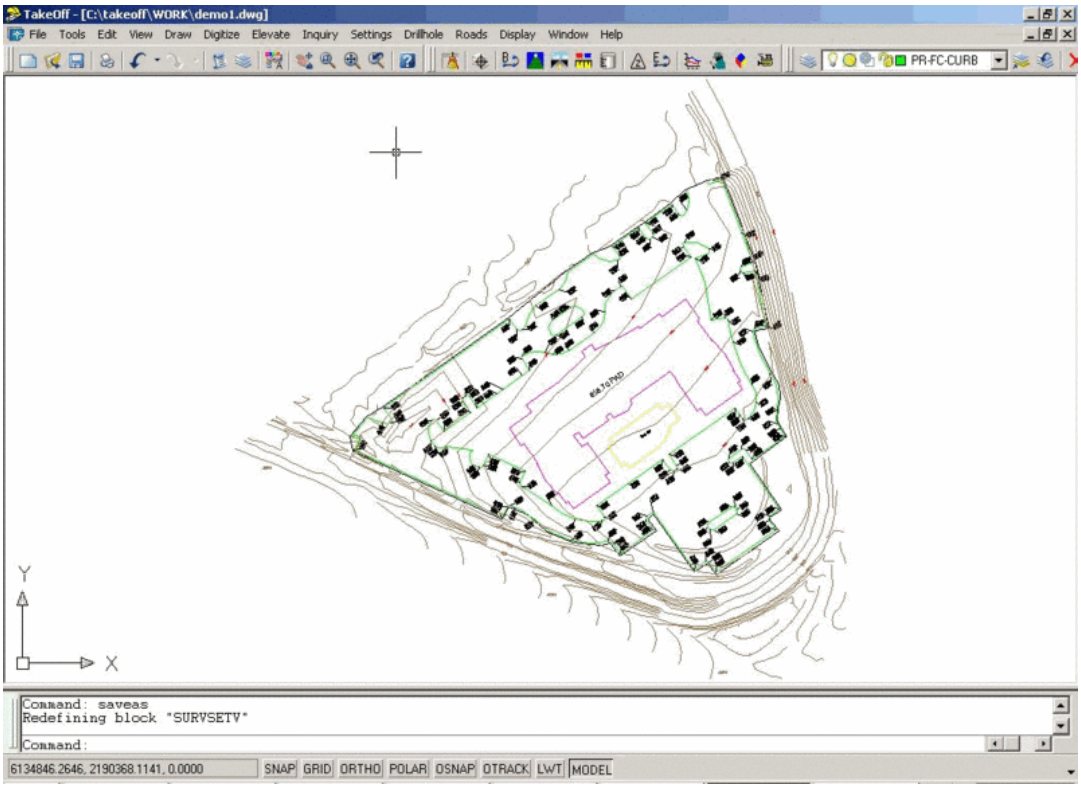
Step 1 (Start Takeoff):

Click the Windows icon for Takeoff to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.

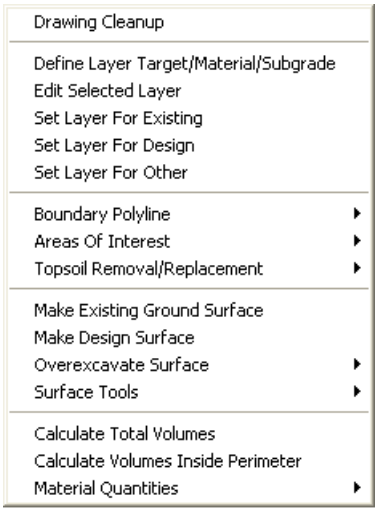


Step 2 (Open Drawing):

From the File menu, choose Open and select DEMO1.dwg from the TakeOff Work folder (ie. C:\Program_files\Carlson_TakeOff_R1\WORK\DEMO1.dwg).



Now we can begin to process this drawing. The main TakeOff commands are listed in processing sequence in the Tools menu. Many of these commands are also grouped as icons in the toolbar shown here.





Step 3 (Drawing Cleanup):

From the Tools menu, choose Drawing Cleanup. Typically, drawings have lots of drafting fixes that must be done before the surfaces can be modeled. This command will apply the selected cleanup functions on the drawing to help automate the cleanup. Here's a brief explaination of the most important of these functions:

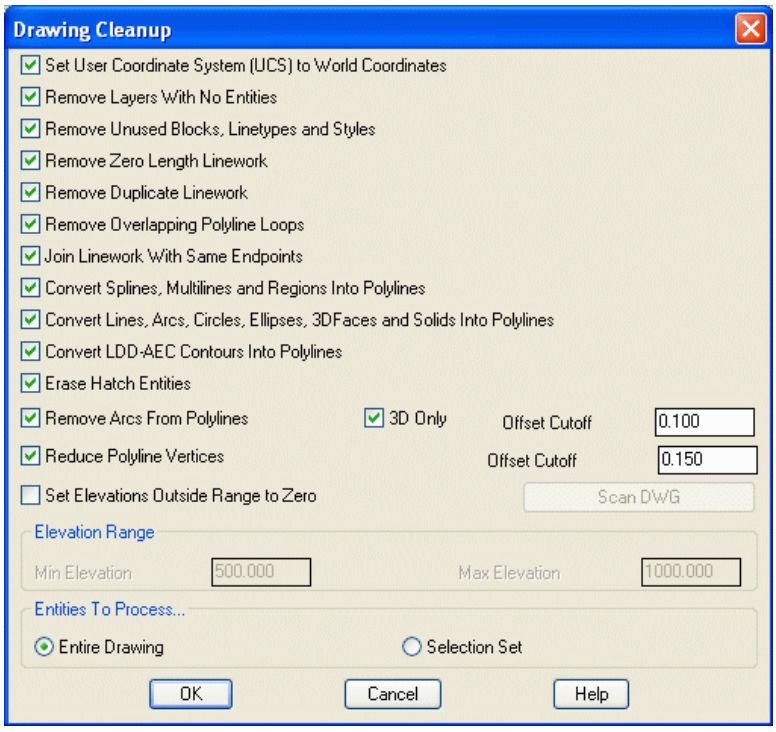
Remove Layers With No Entities: Drawings often have lots of layers. This routine removes layers that have no entities in the drawing so that we don't have to deal with them.

Join Linework With Same Endpoints: This routine will take linework that is broken into multiple segments and join them into a single linework entity. For example, it will join together broken segments of a contour polyline into a single polyline.

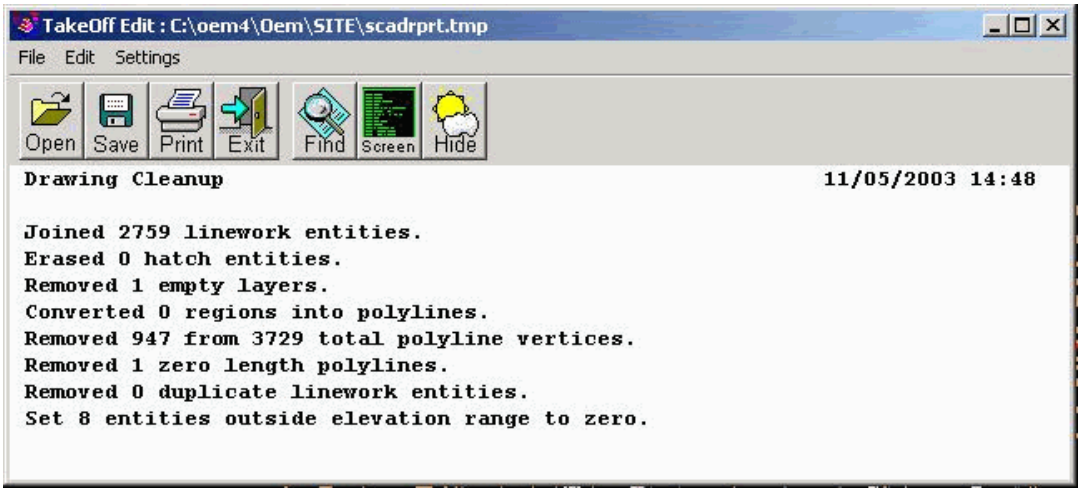
Reduce Polyline Vertices: This routine removes extra vertices from polylines as long as the removing does not shift the polyline more than the specified Offset Cutoff. This will reduce the size and complexity of the drawing.

Set Elevation Outside Range To Zero: In case the drawing contains entities that are outside the range of valid elevations for the site, this routine will set them to zero elevation. The program treats zero elevations as "no elevation" and modeling will filter out these zero elevation entities.

For this site, the elevations are around 800. So let's set the Min elevation to 500 and the Max elevation to 1000. The cleanup will set any entities outside this elevation range to zero. With other TakeOff functions, we can later assign proper elevations to any of these zero elevation entities that need to be used in modeling.



Once the Drawing Cleanup options are set as shown, pick OK. When the cleanup is done, the program will show a report of the cleanup results. Pick the Exit button to exit the report viewer.

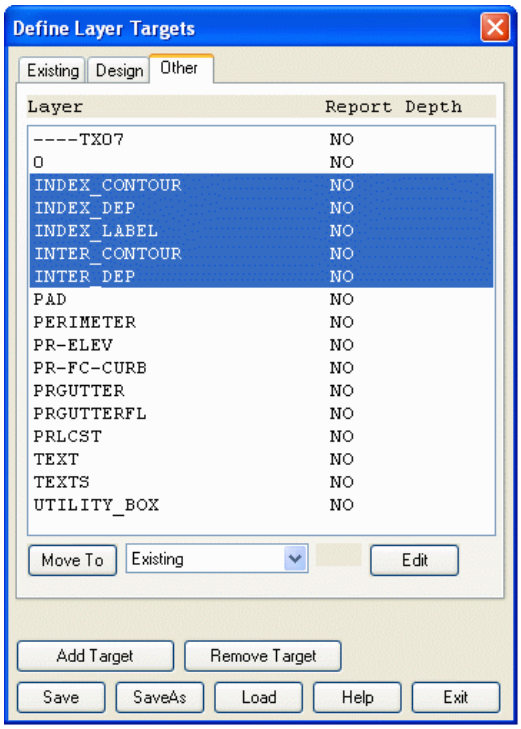

TakeOff Edit : C:\oem4\Oem\SITE\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Drawing Cleanup 11/05/2003 14:48
Joined 2759 linework entities.
Erased 0 hatch entities.
Removed 1 empty layers.
Converted 0 regions into polylines.
Removed 947 from 3729 total polyline vertices.
Removed 1 zero length polylines.
Removed 0 duplicate linework entities.
Set 8 entities outside elevation range to zero.

Step 4 (Layer Targets):

From the Tools menu, choose Define Layer Target/Material/Subgrade. Every entity (line, polyline, point, etc) in the drawing is assigned a layer name. TakeOff uses the entity layer names to define which entities are for the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other.

The Define Layer Targets dialog has three lists of layers: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog.

In this drawing, all the contours are for the existing ground surface. In the layer list, all the layers that start with INDEX and INTER are for these contours. So highlight these layers and then choose Move To Existing. To highlight multiple layers at a time, hold down the keyboard Ctrl key while picking with the mouse.



Next move the layer names that start with "PR" (for proposed) to the Design surface by highlighting these layers and choosing Move To Design. Also move the layer "PAD" to design.

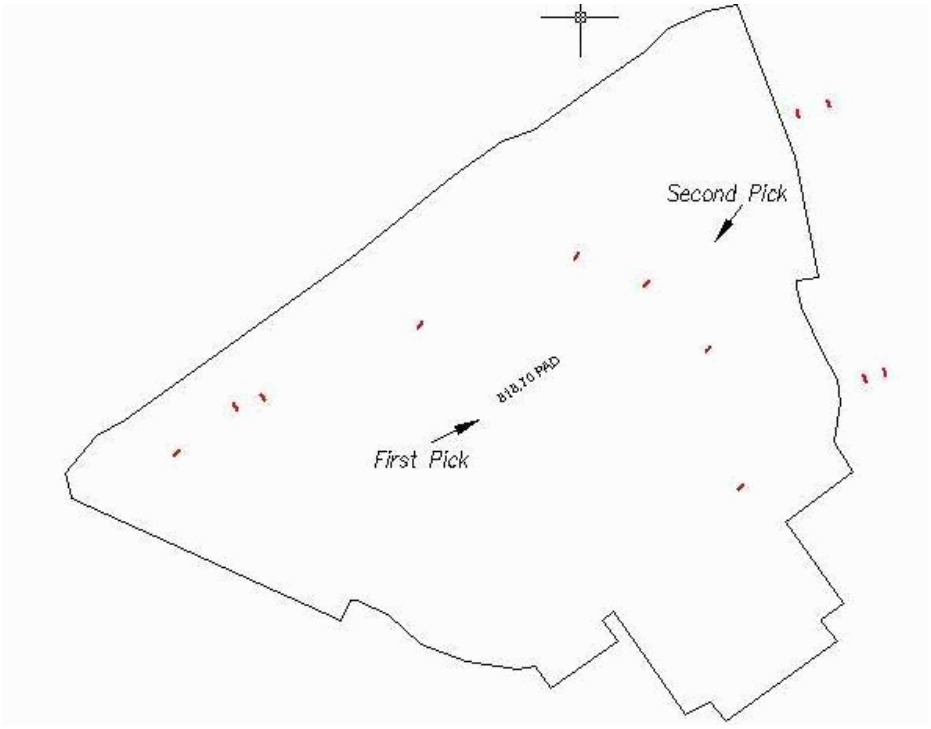
Next pick the Save button to save our changes and then pick Exit.

There are more tools for assigning layer targets. In the Display menu, you can turn on/off whether to display layer targets by using Existing Drawing, Design Drawing and Other Drawing. For example, when Design Drawing is checked, then picking this menu item will uncheck it and turn off all the layers for the design surface. Likewise, picking Design Drawing when it is unchecked will make it checked and turn on the design surface layers.

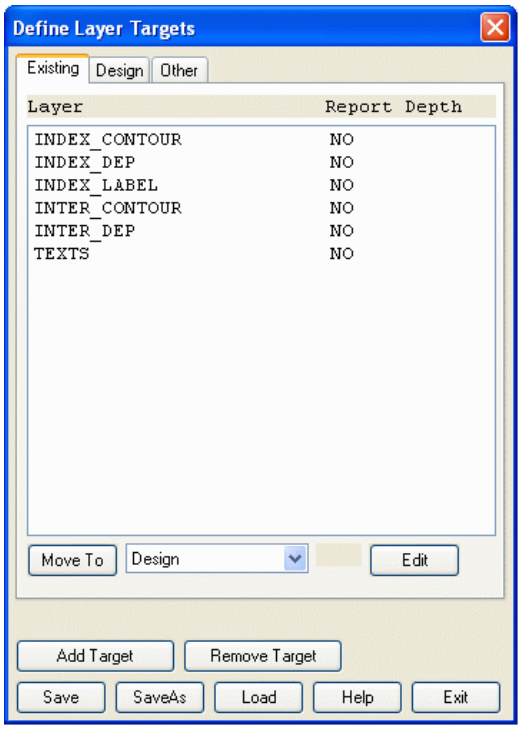
Practice turning on/off the Existing, Design and Other Drawing in the Display menu. When only Existing Drawing is on, you should see just the contours. When only Design Drawing is on, you should see just the design polylines and leader labels. When only Other Drawing is on, you should see the entities that are assigned to neither existing nor design.

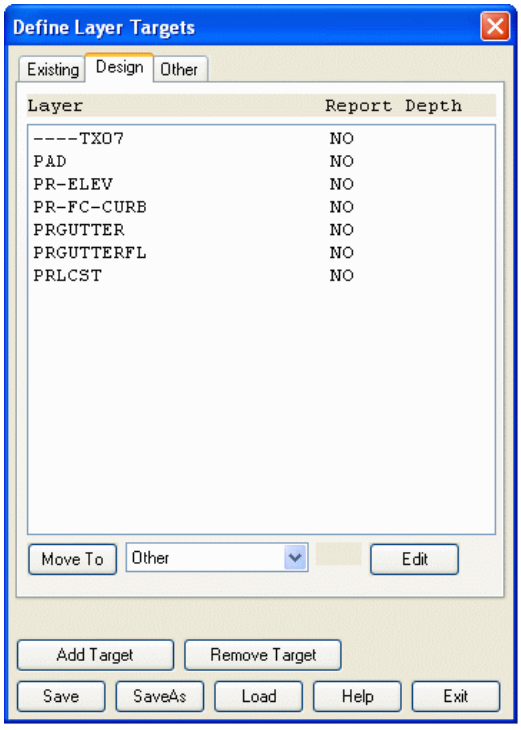


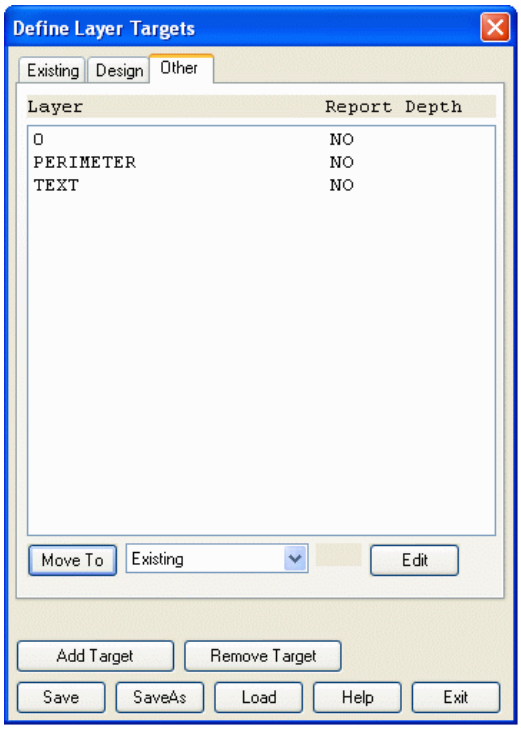
Some of these layers we do want to assign to existing and design. To better see the entities, zoom in on them using the View->Zoom->Window command and pick two points that make a window around the entities as shown. Once zoomed in, you can see a text label of "818.70 PAD" which is for the design surface. Labels "817.00", "818.00" and "819.00" are contour labels for the existing contours. There are a few commands in the Inquiry menu to find out the layer names for these entities: List, Layer ID and Drawing Inspector. Let's run the Layer ID command and pick the "818.70 PAD" label. At the Command line, it reports this layer is "—-TX07". Next pick the "818.00" label and it reports this layer is "TEXTS". Now that we know these layer names, we can return the drawing view back by running View->Zoom->Previous and going to the Display menu and checking on Existing Drawing and Design Drawing.



Next run Define Layer/Material/Subgrade and pick the Other tab. Highlight "—-TX07" and pick Move To Design. Then highlight "TEXTS" and pick Move To Existing.


Define Layer Targets
Existing
Design
Other
Layer
Report Depth
INDEX_CONTOUR NO
INDEX_DEP NO
INDEX_LABEL NO
INTER_CONTOUR NO
INTER_DEP NO
TEXTS NO
Move To
Design
Edit
Add Target
Remove Target
Save
SaveAs
Load
Help
Exit


Define Layer Targets
Existing
Design
Other
Layer
Report Depth
----TXO7 NO
PAD NO
PR-ELEV NO
PR-FC-CURB NO
PRGUTTER NO
PRGUTTERFL NO
PRLCST NO
Move To
Other
Edit
Add Target
Remove Target
Save
SaveAs
Load
Help
Exit



Check that your Layer Targets match the three lists shown here. Then pick Save and Exit.

Step 5 (Define Material/SubGrade):

Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines used for subgrade depth must be closed polylines. TakeOff supports nested subgrade polylines for exclusion areas such as islands by counting how many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade.

First, we need to know the layer names for our subgrades. Go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Then run Inquiry->Layer ID and pick the large pad polyline. It reports that this layer is PAD. Next use Layer ID to pick the curb polyline. It reports that this layer is PR-FC-CURB.

Next we need to make sure that these polylines are closed. In this example, the outside curb polyline is open at the top. To close the polylines, run Edit->Polyline Utilities->Close Polylines. Then pick each of the pad and curb polylines and press Enter when done selecting. Here are the Command line prompts:

Select Polylines to set closed.

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

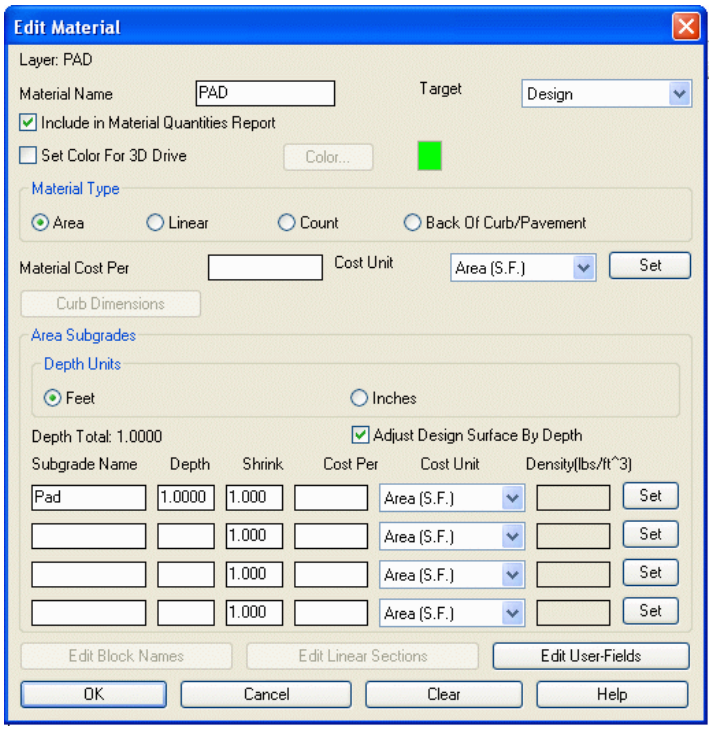
Select objects: 1 found, 6 total

Select objects: (Press Enter)

5 polylines already closed.

Closed 1 polylines.

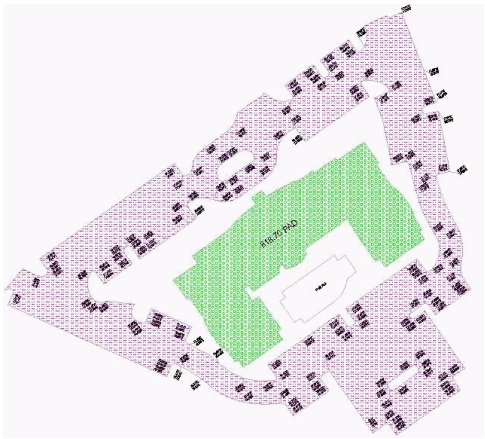
Now run Define Layer Target/Material/Subgrade and pick the Design tab. Highlight layer PAD and pick the Edit button. A dialog appears for defining the pad material properties. Check on the Include In Material Report option, enter the Material Name as "Pad", set the first subgrade name to "Pad", and set the Depth as 1. Once the dialog is filled out as shown, pick OK.



Next pick layer PR-FC-CURB and choose Edit. In the Edit Materials dialog, check on Include In Material Report, set the Material Name to "Pavement", set the first subgrade name to "Pavement", and set the Depth to 1.5. Then pick OK.

To save the subgrade changes, pick the Save button on the Define Layer Targets dialog. Then choose Exit.

Now let's visually verify the subgrade areas. In the Inquiry menu, run Subgrade Areas->Hatch Subgrade Areas. There is a dialog to select which subgrade to hatch. Choose the Pavement. Then there is a dialog for the hatch pattern and color. Click OK. Then run Hatch Subgrade Areas again. This time choose Pad and set the hatch pattern to Hex with green color. The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline. When finished viewing the subgrade areas, run Inquiry->Subgrade Areas->Erase Subgrade Hatches.



Step 6 (Elevate Drawing - 2D to 3D):

TakeOff will model the existing ground and design surfaces based on points, lines and polylines with elevation. It is essential for these drawing entities to have correct elevations in order to get correct surface models. Often the provided drawings will have the drawing entities at elevation of zero with text labels indicating the true elevation. TakeOff has many tools for assigning elevations to these entities.

To help visualize which entities need to be assigned elevation, TakeOff will color entities at zero elevation in grey. As entities get assigned elevation, they return to their original color. This elevation coloring is applied to layers that have been assigned to the existing or design surfaces.

Let's start by working on the existing surface. To isolate the existing entities, go to the Display menu and check on Existing Drawing, uncheck Design Drawing and uncheck Other Drawing. In the Inquiry menu, there are commands for checking elevations. To check the elevation of the contour polylines, run the Inquiry->List Elevation and pick a contour polyline At the command line, it reports the elevation.

Select Entity: pick pad polyline

Elevation: 816.0000

Select Entity (Enter to end): press Enter

In this example, the existing ground surface is defined by just contour polylines and these polylines already have elevation. So there are no changes needed for preparing the existing surface entities. If the contour polylines were at zero elevation, then you could use the Elevate->Assign Contour Elevation commands.

Next let's prepare the design surface. To isolate the design entities, go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Notice that all the design linework is greyed because it is at zero elevation. Run the List Elevation command and pick the main pad polyline. At the command line, it confirms that the elevation is 0.

To set the pad polyline elevation, run Elevate->Set Polyline To Elevation. Enter an elevation of 818.7 (based on the text label). At the Select objects prompt, pick the bigger pad polyline and press Enter.

New Elevation <0.0000>: 818.7

Select Lines, Arcs, Circles or Polylines for elevation change.

Select objects: (pick the pad polyline) 1 found

Select objects: press Enter

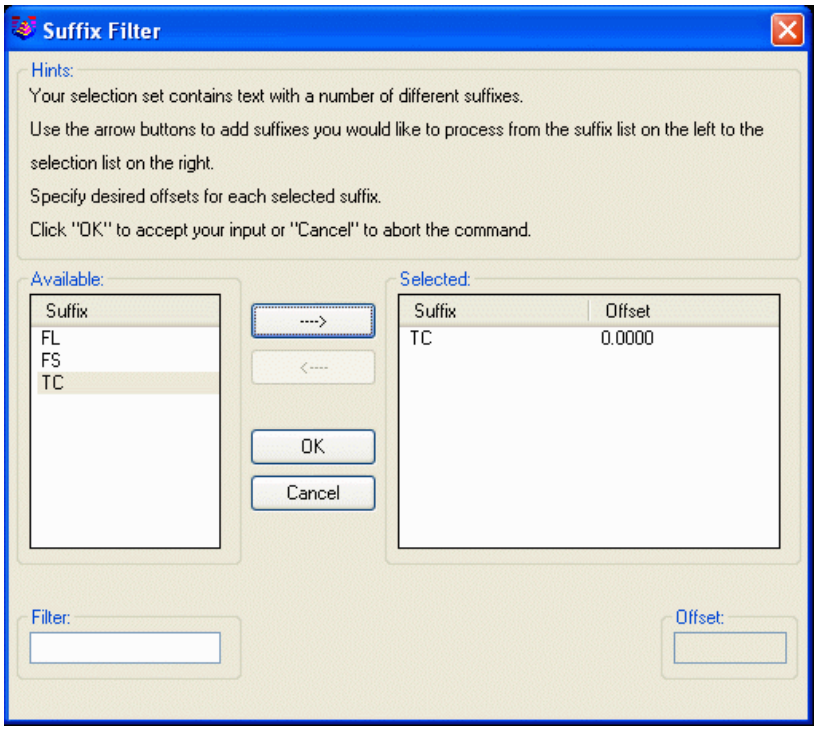
LWPOLYLINE

Number of entities changed> 1

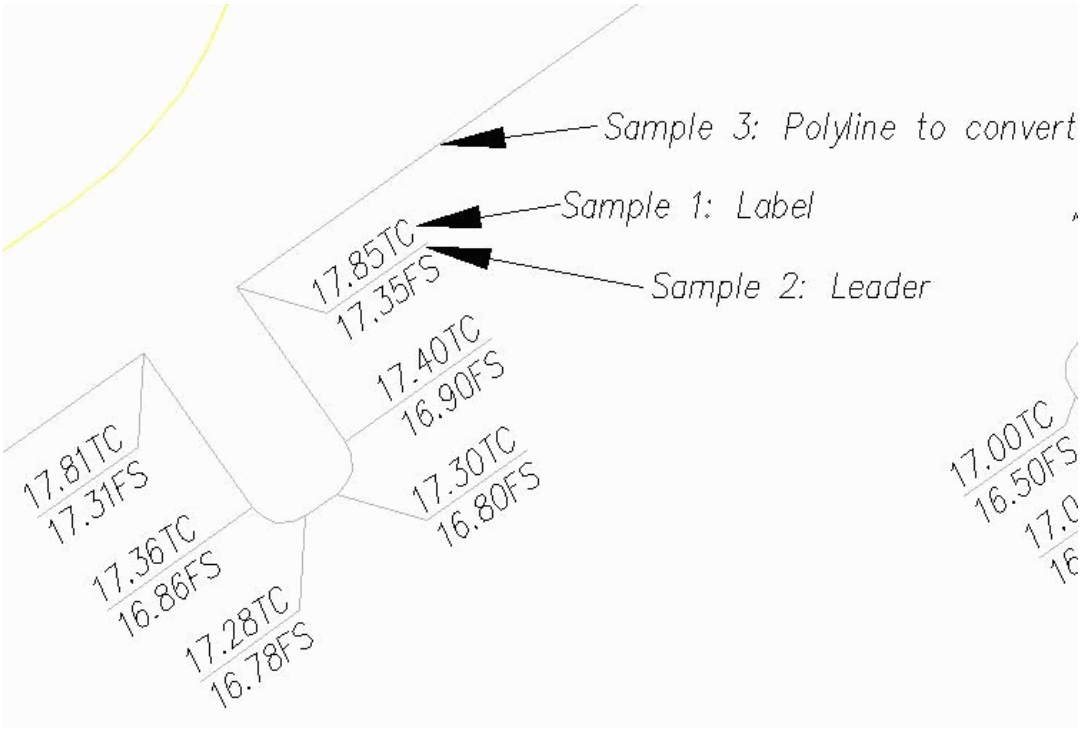
Next let's set the elevation of the smaller pad under the main pad. First, use View->Zoom->Window to zoom in around the smaller pad so that we can read the text label. The label of "17.56" is short for 817.56. In this example, the 800 was dropped from many of the elevation labels to save on label clutter. Run Set Polyline To Elevation again. This time enter an elevation of 817.56 and pick the smaller pad polyline. Then run View->Zoom->Previous to get back to the full view of the site.



Finally, we need to set the elevations for the curb polylines. First, use View->Zoom->Window to zoom in around some of the curb labels below the smaller pad. Then run Elevate->2D to 3D Polyline->Text With Leader. This command will assign the elevations from the labels to the polylines by following the label leader to find the position on the polyline. For polyline vertices without elevation labels, the elevations will be interpolated from the other labels. Before processing, this routine prompts for samples of the elevation label, the leader and the polylines to convert. Then you can select all the entities in the drawing and the routine will sort the labels, leaders and polyline by the sample layers and assign the elevations. For this example, pick one of the labels with a "TC" suffix as the elevation text sample. Then pick the leader line for the annotation leader sample. Then pick the curb polyline for the polyline to convert sample. At the Select objects prompt for processing, type "all" to select all the drawing entities and press Enter. For the elevation to add, enter 800 so that labels like "17.81" get assigned as 817.81.



Next a dialog appears for selecting which labels names to use. When TakeOff detects different text labels within the elevation labels, you need to choose which ones to process. In this case, we only want the labels with "TC". So highlight TC, pick Add and then pick OK.



Select sample of elevation text: pick label

Select sample of an annotation leader: pick leader line

Select sample of a polyline to convert: pick curb polyline

Select polylines to convert, leaders and elevation labels to process.

Select objects: all

Select objects: press Enter

Joining adjacent polylines...

Reading the selection set ...

Enter elevation to add to label values <0.00>: 800

Pre-processing entity #420 of 420

Processing leader #141

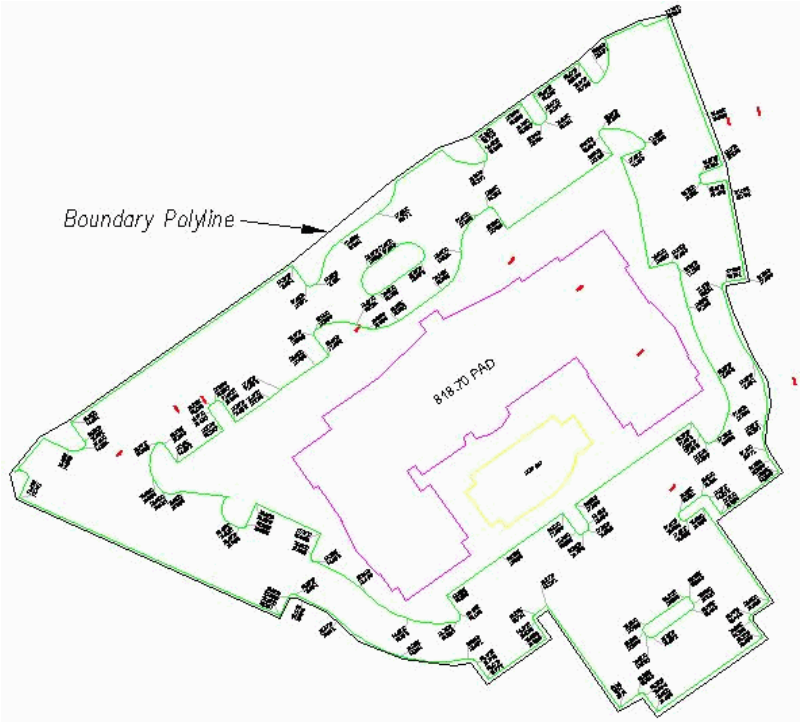
Remaking polyline #4

All the curb polylines now have elevations.

Now run View->Zoom->Previous to return to the full site view. The design polylines should now have colors because the elevations are assigned.

Step 7 (Boundary Polyline):

The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. In this example, there is a closed polyline on the PERIMETER layer. The layer target for this layer is Other. Go to the Display menu and check on Other Drawing so that the perimeter is displayed. Then run Tools->Boundary Polyline->Set Boundary Polyline and pick the perimeter polyline. This selected polyline is now set as the boundary polyline for the rest of the TakeOff routines.



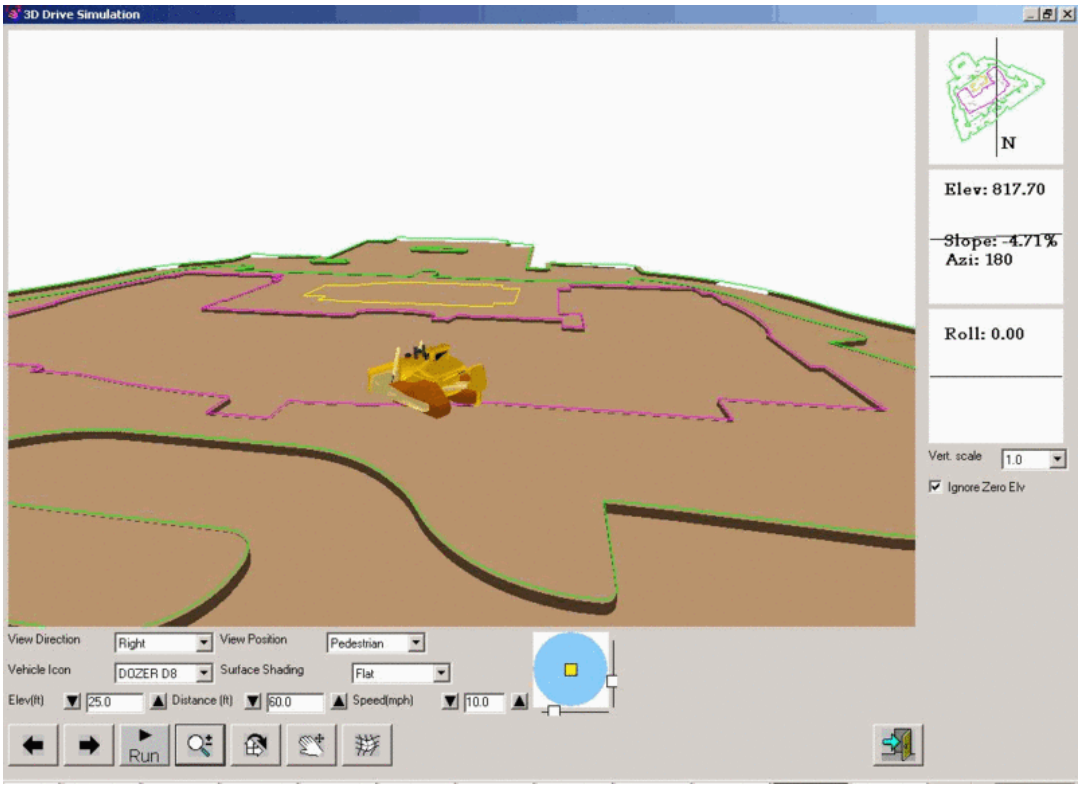
Step 8 (Model Existing and Design Surfaces):

To calculate volumes, TakeOff needs two surfaces: existing ground and design. These surfaces are modeled by

triangulation. With the preparation of the previous steps, we're now ready to make the models. The drawing entities have been cleaned up, assigned elevations and assigned target surfaces by layer. Making the models is now a one step process. To make the existing ground surface, run Tools->Make Existing Ground Surface. The program will process the entities and make the triangulation surface. Then to make the design surface, run Tools->Make Design Surface.

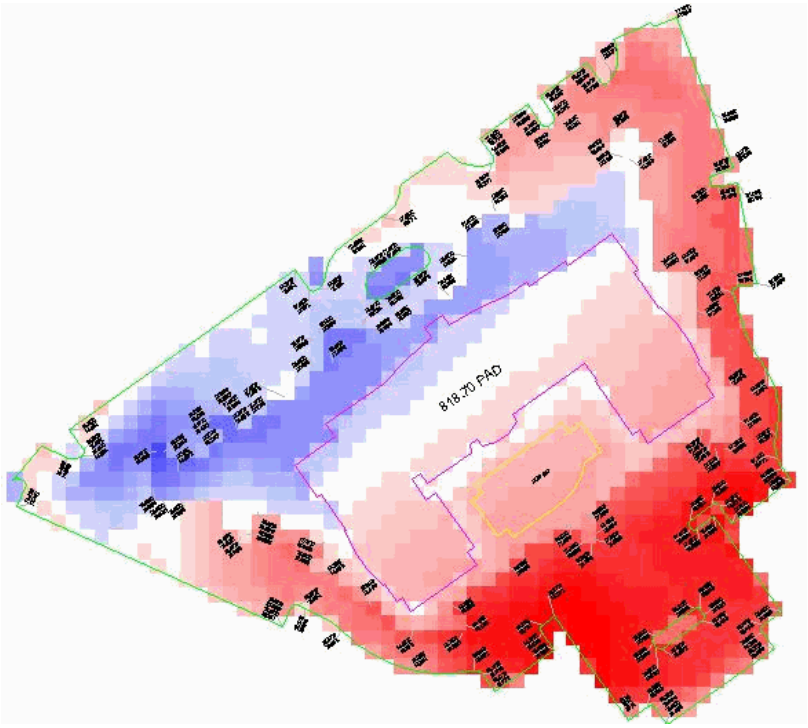
Step 9 (3D Drive Simulation):

As a visual check that the design surface modeled correctly, let's run the View->3D Drive Simulation command. This routine shows a 3D view of the site and allows you to drive around. This is a good way to check that the surface modeled correctly. We want to make sure that there are no elevation spikes and that the subgrade depths are modeled. To drive the site, choose a View Direction, View Position and Vehicle. Then pick the Run button and use the arrow keys to turn. Pick the Stop button to pause the moving. You can also try the Surface Shading options for different views of the surface. When done with the 3D Drive Simulation, pick the Exit button (Arrow with door image).



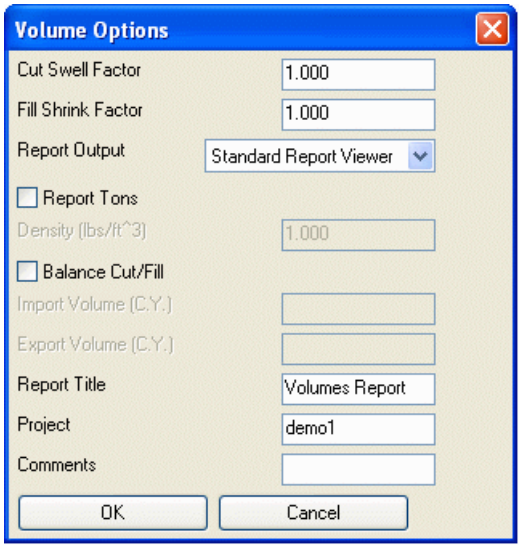
Step 10 (Cut/Fill Color Map):

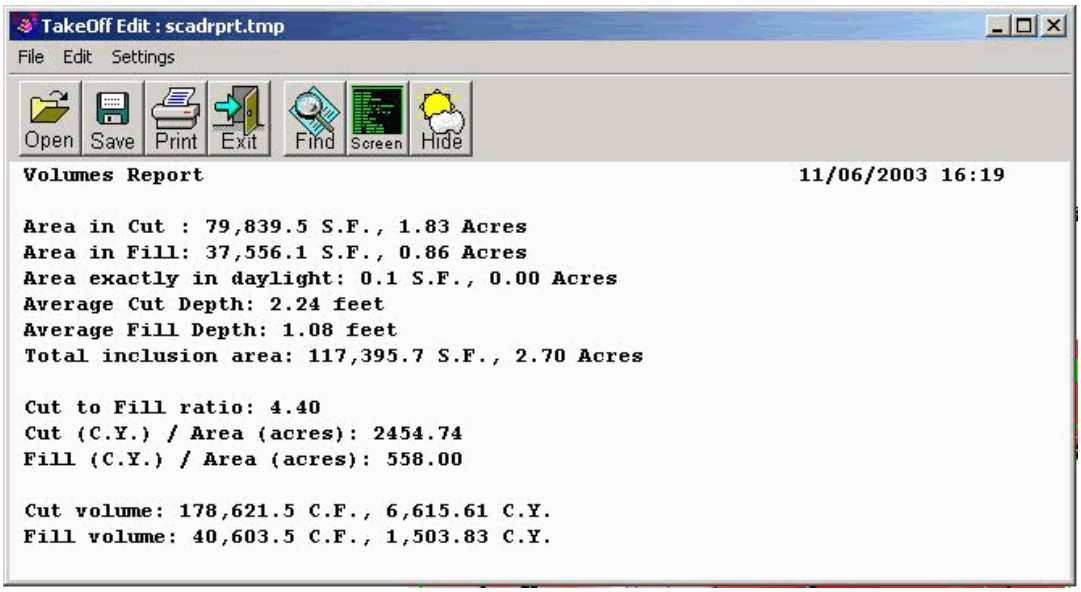
Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as another check that the models are correct. In the Display menu, choose Cut/Fill Color Map. Cut areas are drawn in different shades of red for different depths of cut while fill areas are drawn in blue. To change the resolution of the color blocks, run Display->Display Options and change the Cut/Fill Color Map Subdivisions. This parameter is the number of rows and columns of color blocks to create. To turn off the color map, go to the Display menu and pick Cut/Fill Color Map to uncheck it.



Step 11 (Calculate Volumes):

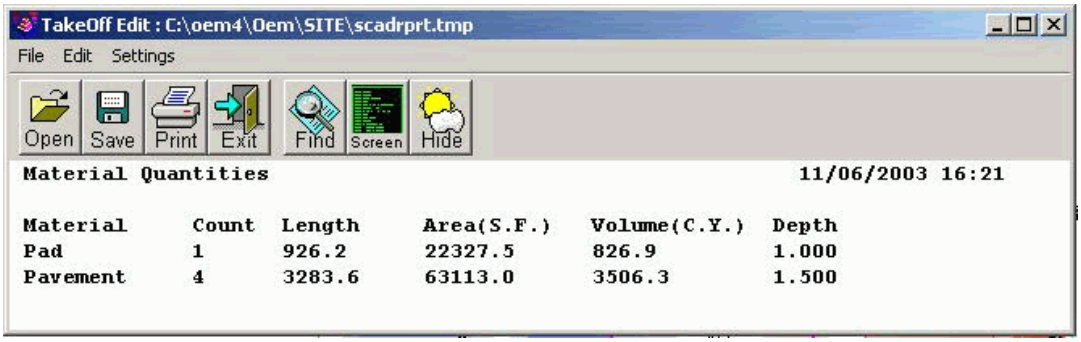
To calculate volumes, run the Tools->Calculate Total Volumes command. There is an options dialog for setting the cut swell factor and fill shrink factor. These values get multipled into the cut/fill volumes. Set these factors as desired and click OK. Then the routine calculates the volumes and display the report which includes the cut/fill volumes and areas. The report can be printed or saved to a file. Pick the Exit button to exit the report viewer.




TakeOff Edit : scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 11/06/2003 16:19
Area in Cut : 79,839.5 S.F., 1.83 Acres
Area in Fill: 37,556.1 S.F., 0.86 Acres
Area exactly in daylight: 0.1 S.F., 0.00 Acres
Average Cut Depth: 2.24 feet
Average Fill Depth: 1.08 feet
Total inclusion area: 117,395.7 S.F., 2.70 Acres
Cut to Fill ratio: 4.40
Cut (C.Y.) / Area (acres): 2454.74
Fill (C.Y.) / Area (acres): 558.00
Cut volume: 178,621.5 C.F., 6,615.61 C.Y.
Fill volume: 40,603.5 C.F., 1,503.83 C.Y.

Step 12 (Material Quantities):

To report the quantities, run the Tools->Material Quantities->Standard Report routine. The report includes the count, length, area and volume for each type of material that was assigned for reporting in the Define Layer Target/Material/Subgrade command. The Material Quantities->Custom Report routine can be used to reporting these values with control of the report format and the option to export to Excel.


TakeOff Edit : C:\oem4\Oem\SITE\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Material Quantities 11/06/2003 16:21
Material Count Length Area(S.F.) Volume(C.Y.) Depth
Pad 1 926.2 22327.5 826.9 1.000
Pavement 4 3283.6 63113.0 3506.3 1.500

Road Work

This lesson takes a drawing file through the steps of road design.

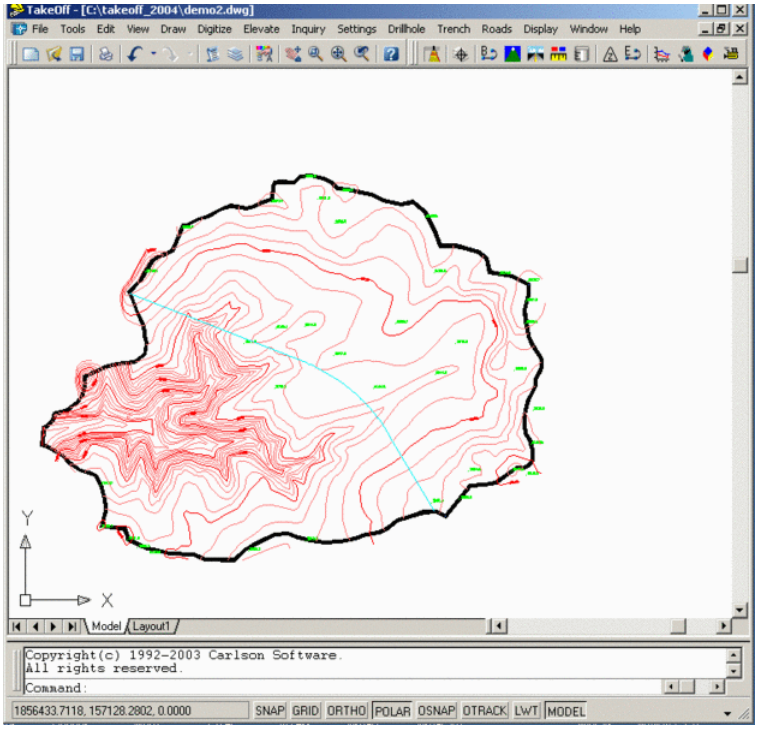
Step 1 (Start Takeoff):

Click the Windows icon for Takeoff to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.



Step 2 (Open Drawing):

From the File menu, choose Open and select DEMO2.dwg from the Takeoff Work folder (ie. C:\Program_files\Carlson_Takeoff_2004\WORK\DEMO2.dwg).

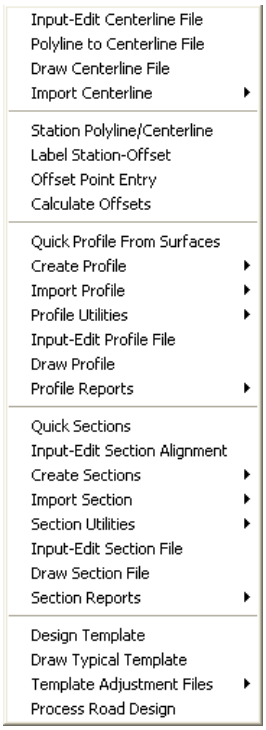

TakeOff - [C:\takeoff_2004\demo2.dwg]
File Tools Edit View Draw Digitize Elevate Inquiry Settings Drillhole Trench Roads Display Window Help
Y
X
Model Layout1
Copyright(c) 1992-2003 Carlson Software.
All rights reserved.
Command:
1856433.7118, 157128.2802, 0.0000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL

Step 3 (Existing Ground Surface):

For the road cut/fill slopes to be created, it needs the existing ground surface to tie into. First we need to define the layers of the existing ground surface. Run Tools-> Set Layer For Existing, pick on both a light-lined and heavy-lined contour, a spot elevation, and press Enter. This will set the layers CTR, CTRINDEX, and TO-PREM-SPOT-PNT to the existing surface. The limits of the site are defined by a closed polyline. Run Tools->Boundary Polyline->Set Boundary Polyline and pick the black polyline around the perimeter. This selected polyline is now set as the boundary polyline for the existing surface. To make the existing ground surface, run Tools->Make Existing Ground Surface.

Step 4 (Creating a Centerline):

Now we will use commands under the Roads menu.



Notice that the Roads menu is broken down into four sections: Centerline commands, Profile commands, Section commands, and Template commands. A road needs design input from all four sections to be created. It does not matter in what order the commands are run, but in this example we will run the commands in descending order.

A centerline file is necessary for the final road design routine. We will do the simplest variation, which is picking a polyline. There are other methods to design a centerline, and they are documented in the manual. Go to Polyline to Centerline File in the first grouping of commands under Roads. A file selection dialog will appear. Enter a centerline file name of demo2.cl and pick save. Follow the prompting:

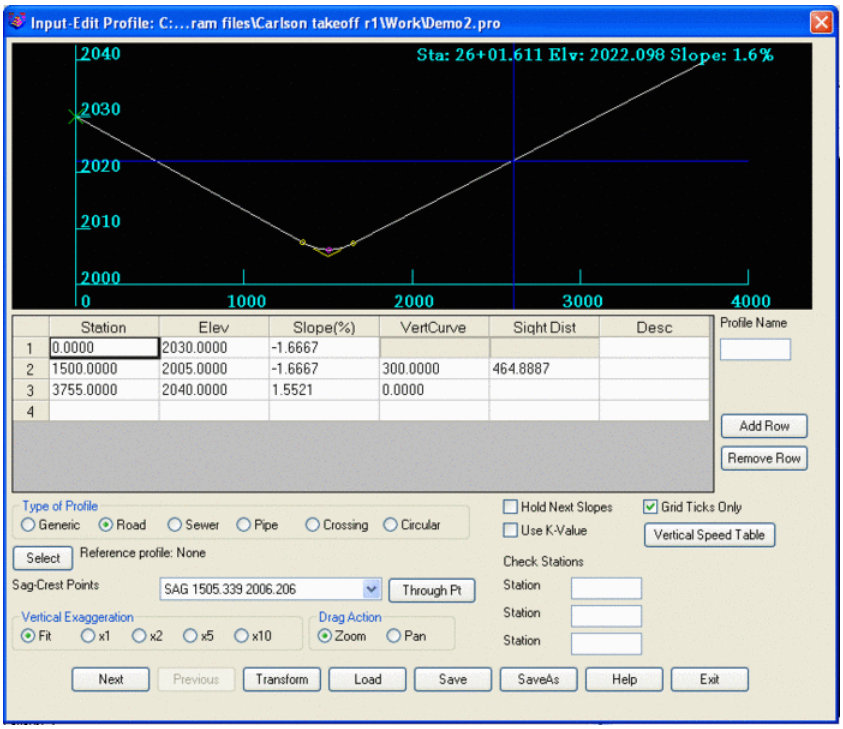
Beginning Station <0+00>: Press Enter
Polyline should have been drawn in direction of increasing stations.
Select polyline that represents centerline: Select the polyline that crosses the middle of the site with the layer name CLINE.
Station North(y) East(x) Description

0.0000 159718.2034 1857460.9166 LI
1798.2055 159058.6315 1859133.7908 PC
2912.2263 158347.2903 1859964.4134 LI
3755.6840 157619.5351 1860390.7855 LI
Press ENTER to continue.

Your Command Line should have the same values as these, as they are from the same line. Hit the F2 key or press Enter to return to the main screen.

Step 5 (Input-Edit Profile):

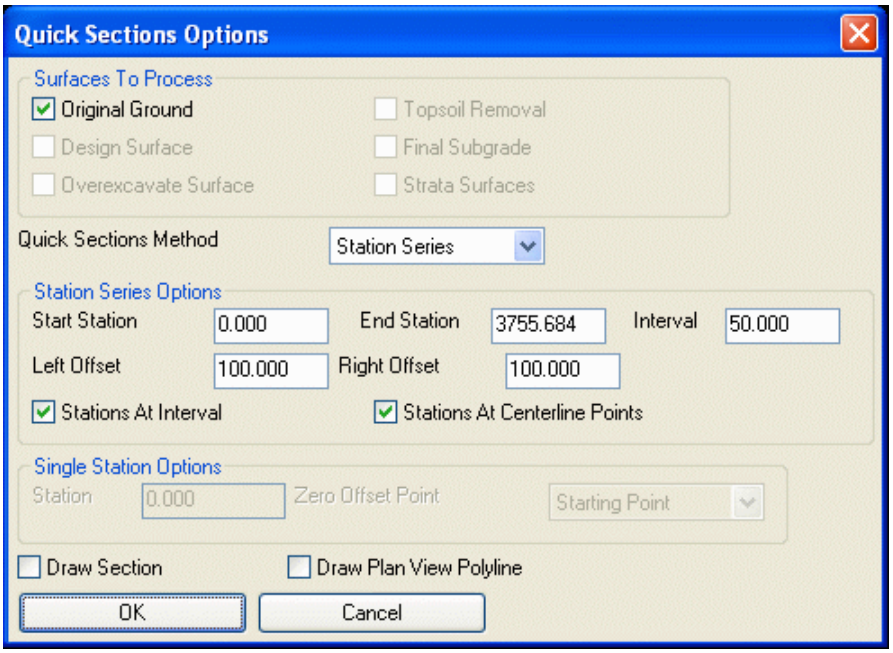
In this routine we will create a Profile file. There are different ways to define a road profile. In this case we will enter values from a given design. Go to Roads-> Input-Edit Profile File, create a new file, and name it Roaddemo. The Input-Edit Profile window will be displayed. Under Type of Profile select Road from the dialog box. In the spreadsheet, you can add design features for the Profile of the road. In this example, enter in the stations 0.0, 1500.0, and 3755.6840, with elevations of 2030, 2005, and 2040. Next, set the Vertical Curve for the middle station at 300.0.



The Slope Percentages and the Sight Distances are automatically computed for you. Select Save and Exit.

Step 6 (Quick Section):

Now we will create the cross section file (*.SCT). The cross sections define the existing ground for the road to tie into. Run Roads->Quick Section, fill the dialog as shown, and click OK. Your Offsets should be far enough away to tie in the Cut/Fill slopes.



Next, you will be prompted to load a centerline file or to to pick a polyline from the screen. Type C for Centerline and select the demo2.cl file we created earlier. Your section file is now created, your Command line should read as follows:

Command: quicksct

Pick starting point (CL-Centerline,P-Polyline): c

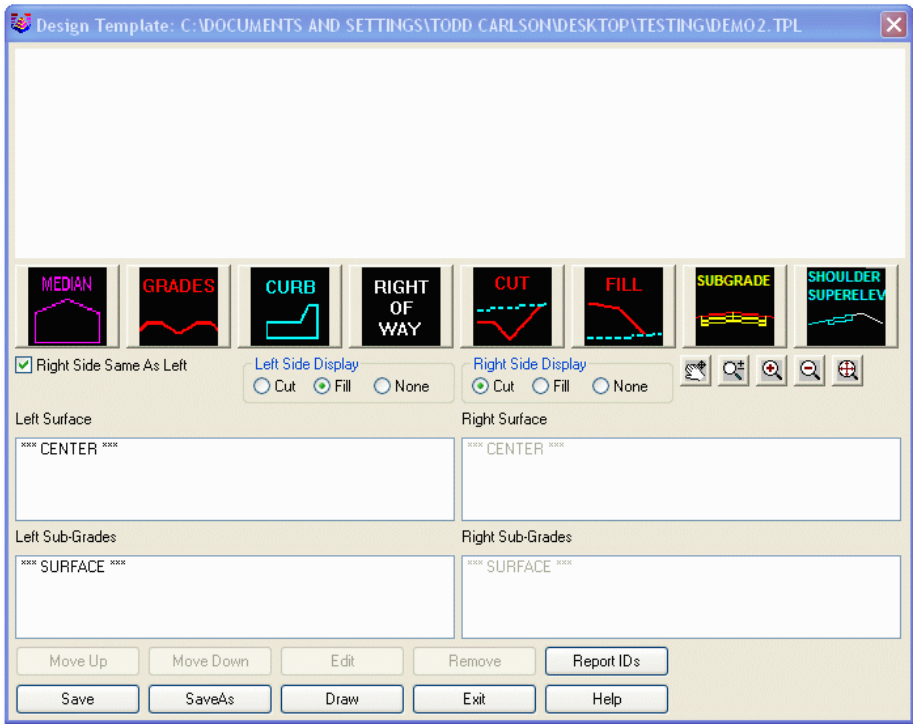
Polyline should have been drawn in direction of increasing stations.
Loading edges...
Loaded 4244 points and 12207 edges
Created 7964 triangles

Writing section C:\Takeoff_2004\demo2a-og.sct

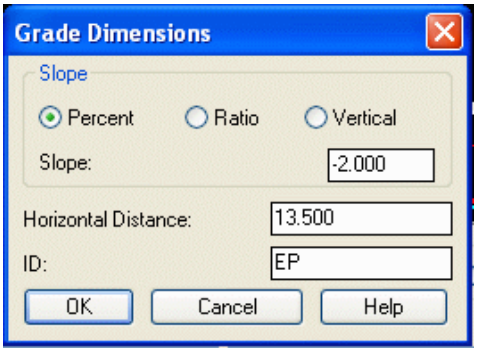
Step 7 (Design Template):

Let's design a wide boulevard with curb and gutter on the outside. The cut slope will be 2:1. In fill, the condition will be 3:1 in all fill under 6' and 2:1 in all fill over 6' in depth. Pavement depths will be 4" of asphalt.

First, select Design Template under Roads, name the .TPL file demo2, and open it. A dialog appears where you enter segments of the template. We will enter a symmetrical template, with 13.5' pavement sections on either side of the centerline, connecting to a 2' curb and gutter, with 18" of gutter and 6" of curb. Then we'll add a 6' shoulder.

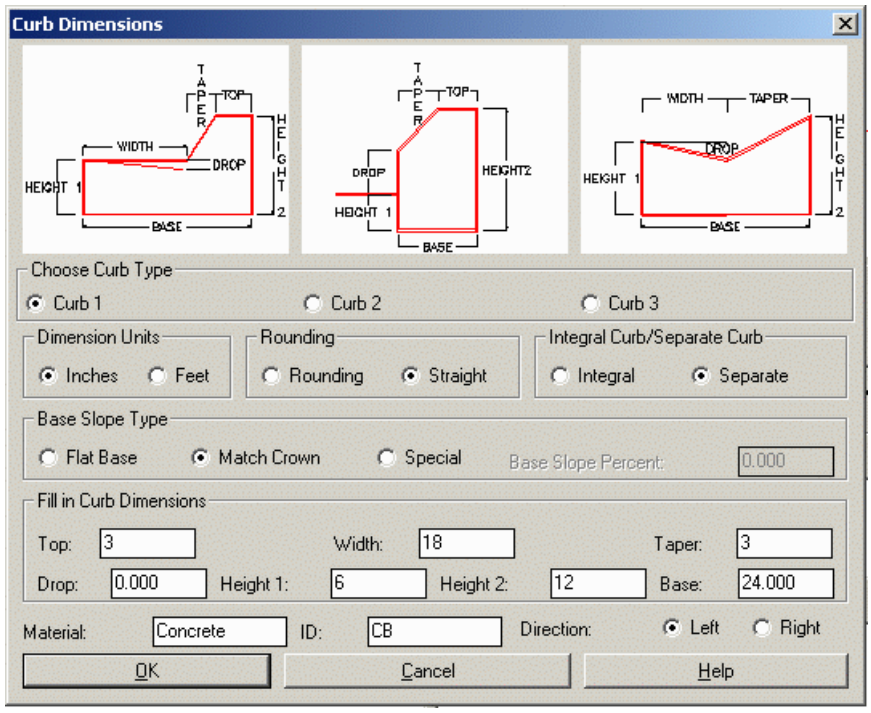


For the lanes, click the Grades Icon.



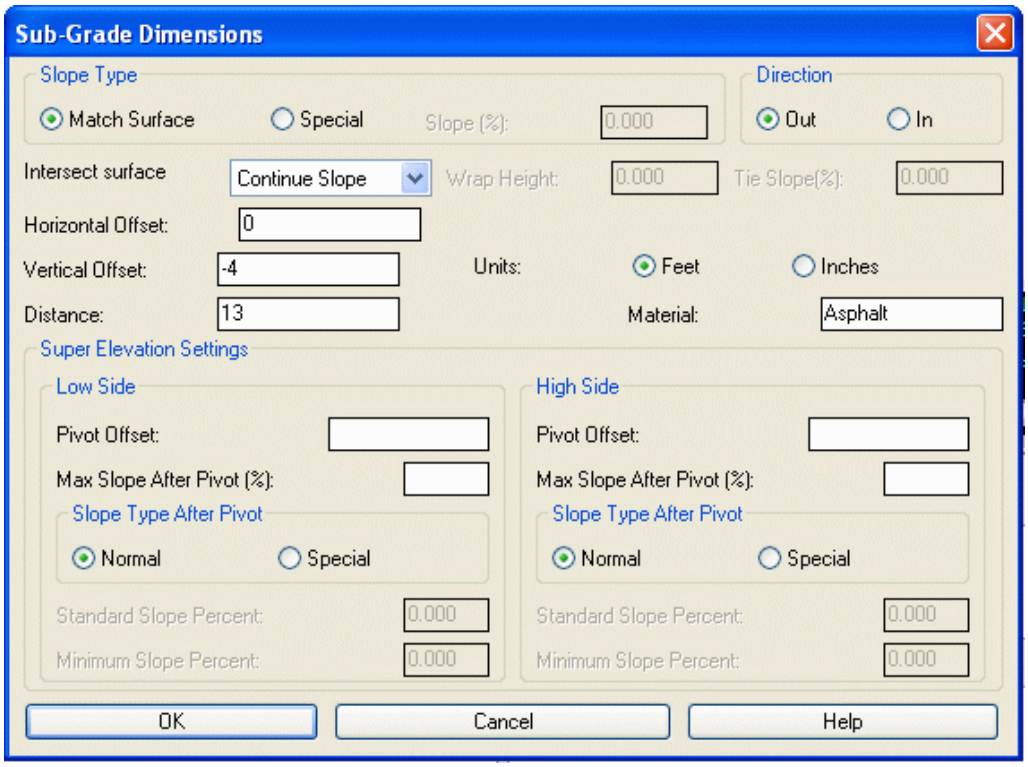
The above 'child' dialog is shown, enter in: Slope: -2, Horizontal Distance: 13.5, and ID: EP. Click OK. You'll note that the lanes draw in the little preview window.

Click on the Curb Icon. Fill out as shown below and click OK.

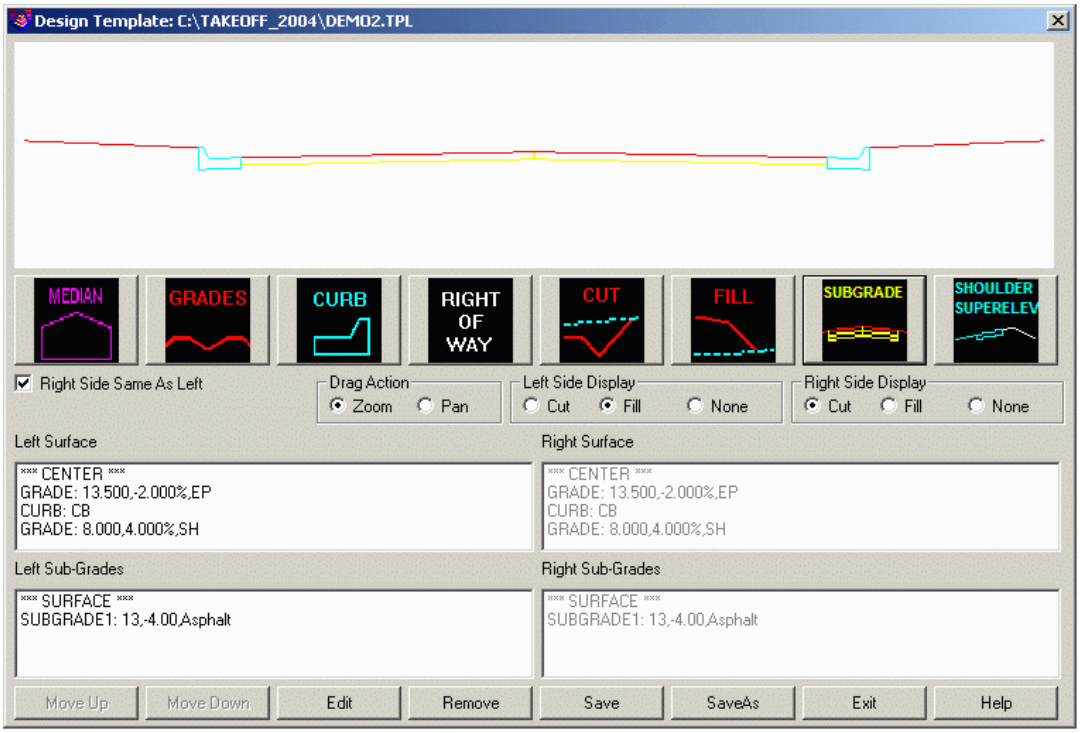


Next up, we will add a shoulder, going up hill at 4% for 8'. Click on Grades again and enter in a slope of 4, a Horizontal Distance of 8, and the ID as SH.

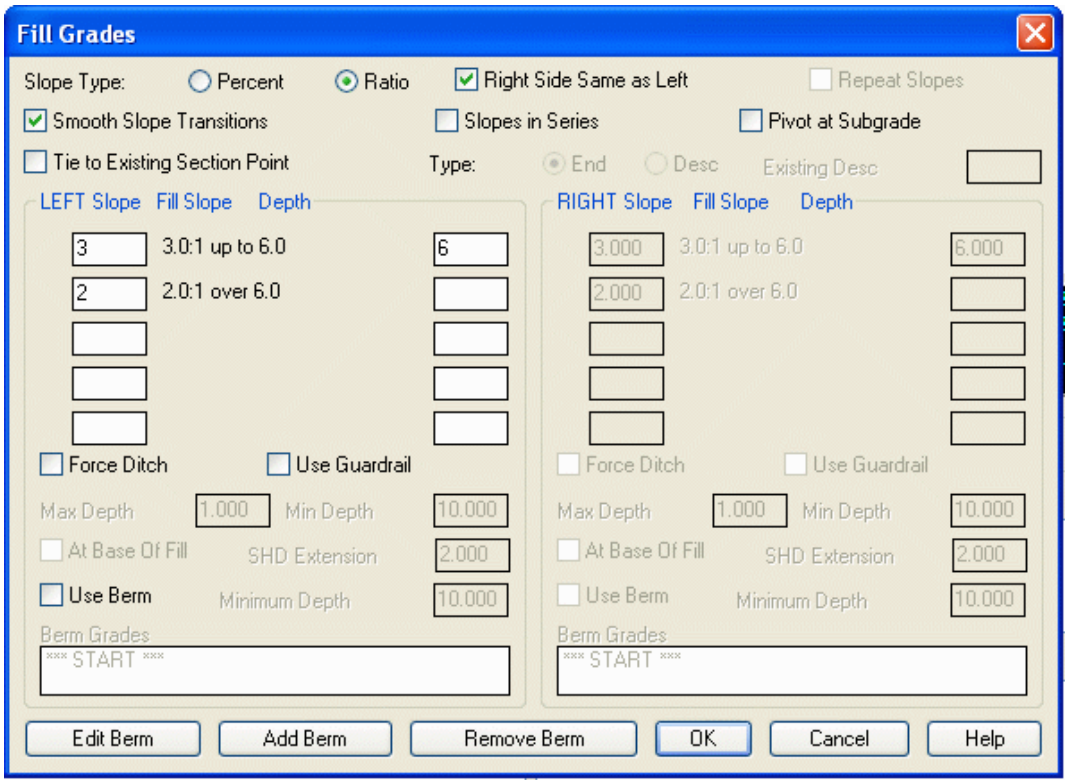
We are now finished with the surface and can set subgrade. Select the Subgrade icon, second from the right (yellow color). We will create an asphalt subgrade which will run straight out and hit the curb.

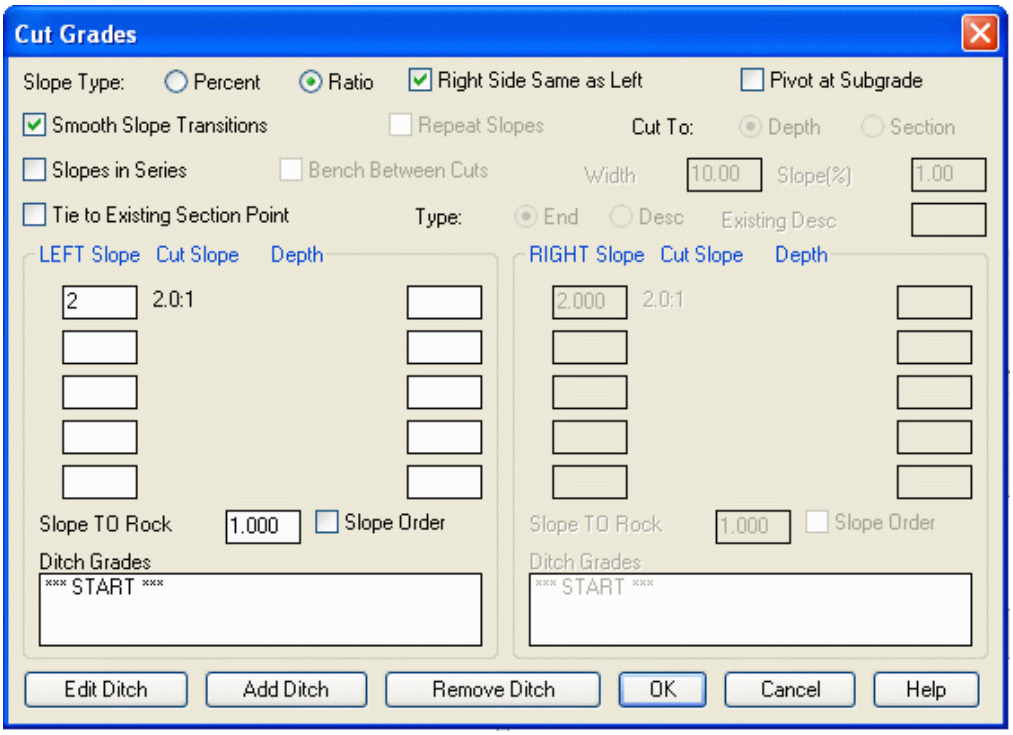


Complete as shown and click OK. Here's what our template looks like so far:



Now we will add the outslope conditions. They are done with the Cut and Fill icons. Click on Fill and make three entries: under LEFT Slope enter in 3 (for 3:1), under Depth enter 6 (up to 6'), then again under LEFT Slope enter in 2 (for 2:1 over 6'). Select OK and click the icon for Cut. Just one entry here: under LEFT Slope enter in 2 (for 2:1 normal cut). Click OK.

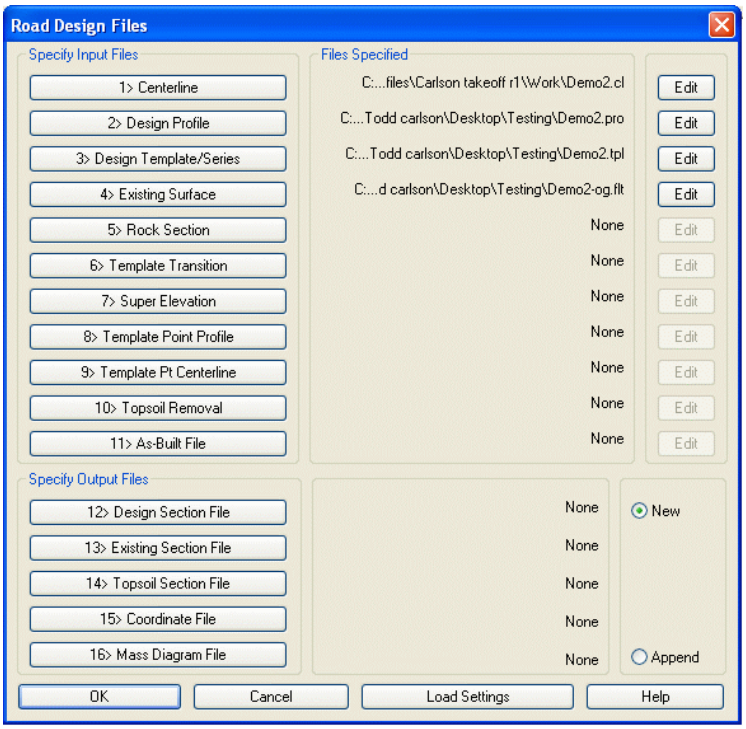




Now click Save. The template is complete.

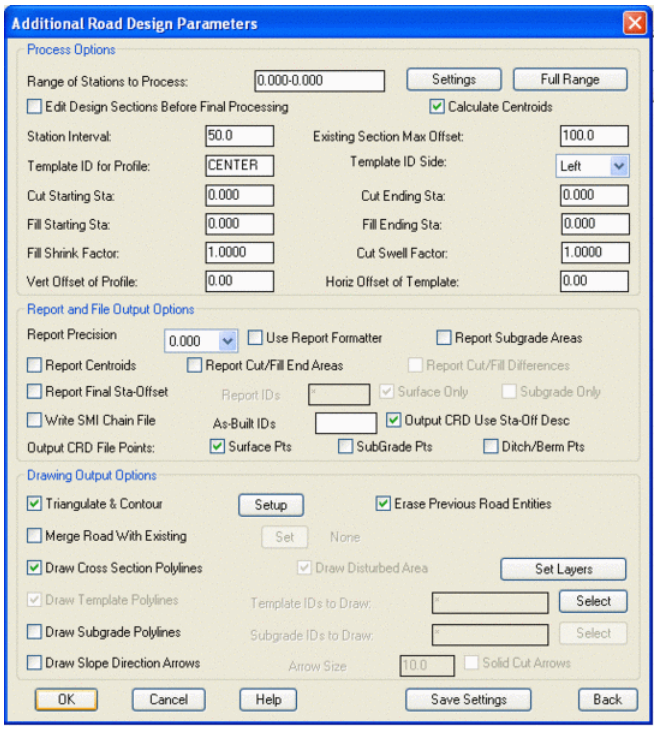
Step 8 (Process Road Design):

This is the routine that weaves everything together. Select Process Road Design, the last command in the Roads menu. The Specify Input Files column on the left allows you to choose the files to be used in the road design. We have already created the four needed files, go ahead and select them now and then click OK.



In the next dialog you can select different output features. For 3D viewing in the next step, toggle on Triangulate & Contour . It can be found in the bottom left portion of the Additional Earth Works Parameters dialog. Click OK.

Note: To generate a plot of road sections, specify an output section file in the 1st Road Design dialog. Then run Draw Sections in the Roads menu.



The following report listing the total Cut, Fill, Subgrade, and Curb volumes.

```
Process Road Design                                    07/29/2004 13:51

Template File> C:\TAKEOFF_2004\DEMO2.TPL
Profile File> C:\TAKEOFF_2004\ROADDEMO.PRO
Existing Surface File> C:\takeoff_2004\demo2a-og.sct
Centerline File> C:\takeoff_2004\demo2.cl

Processing 0+00.000 to 37+55.680
Total Cut : 134177.962 C.F., 4969.554 C.Y.
Total Fill: 2103572.986 C.F., 77910.111 C.Y.
Total SUBGRADE1 - Asphalt: 25357.092 C.F., 939.152 C.Y.
Total CURB - Concrete: 11291.988 C.F., 418.222 C.Y.
```

Trim existing contours inside disturbed area [Yes/<No>]? Press enter to say no

In the Contour Options dialog change the layer name to FINAL ROAD and make the contour interval 2. Click OK and your Road is complete! Here are the Road Design prompts:

Command: eworks

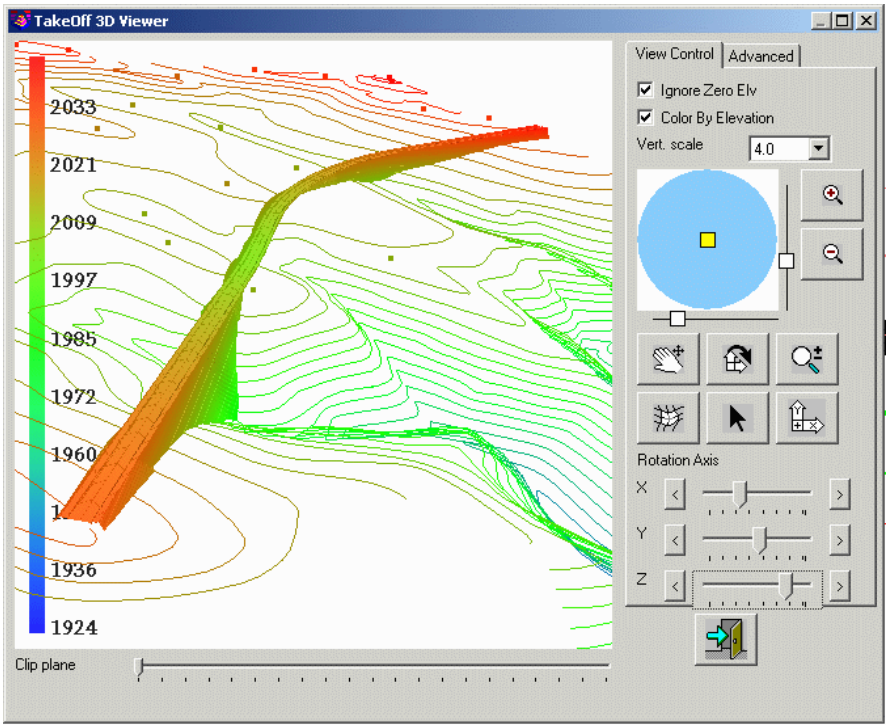
Initializing EarthWorks ...
Processing station: 3755.680
Drawing offset 3D polylines: TIE
Calculating volumes ...
Trim existing contours inside disturbed area [Yes/<No>]? <Enter>

Reading points... 2631
Inserted 2631 points
Inserted 2618 breakline segments

Drawing Triangulation 3D Faces ...
Contouring elevation 2040
Inserted 811 contour vertices.

Step 9 (3D Viewer):

Now that your Road is complete lets view it in 3D. Go to View, 3D Viewer Window, type in ALL, and press enter.



Here is our road with a Vertical Scale of 4. Color By Elevation has also been toggled on. Use the X, Y, and Z control bars at the bottom to rotate the drawing in the 3D viewer.

Trench Network

This lesson takes a drawing file through the steps of trench network quantities.

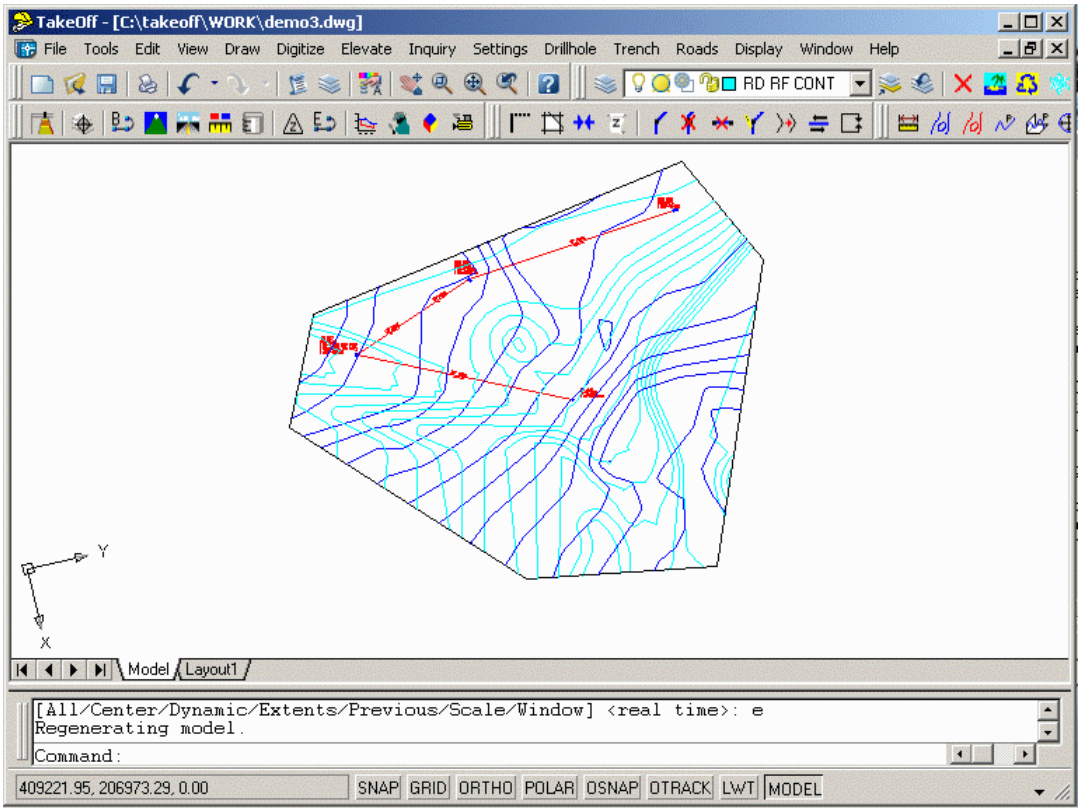
Step 1 (Start Takeoff):

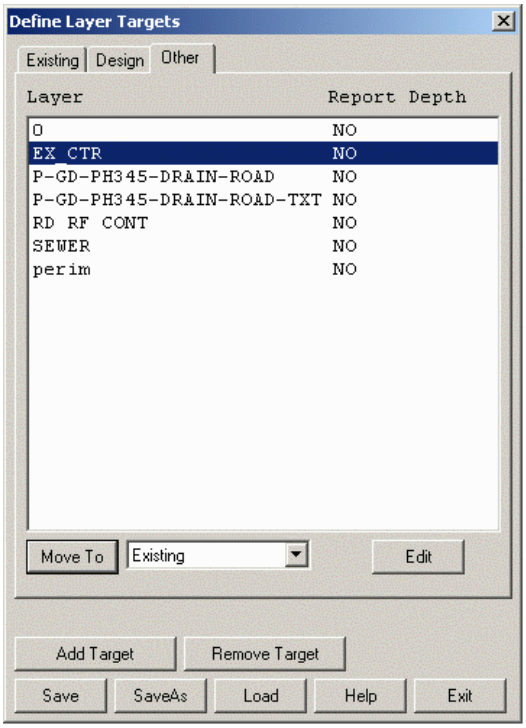
Click the Windows icon for Takeoff to launch the program. You may be presented with a "Startup Wizard" dialog and if so, click Exit.



Step 2 (Open Drawing):

From the File menu, choose Open and select DEMO3.dwg from the Takeoff Work folder (ie. C:\Program_files\Carlson_Takeoff_2004\WORK\DEMO3.dwg).





Step 3 (Make Existing and Design Surfaces):

In order to calculate trench quantities and profiles, we need surfaces for the existing ground and design.

First we need to define the layers of the surfaces. Run Tools->Define Layer Target/Material/Subgrade. Then from the tab labeled "Other", highlight "EX_CTR" from the layer list, pick "Existing" from the Move To list and pick the Move To button. Next, highlight "RD RF CONT", choose "Design" from the Move To list and pick the Move To button. Now choose the Save and then Exit buttons. This assigned layer "EX_CTR" to the existing ground surface and "RD RF CONT" to the design surface.

Next, let's set the site perimeter. Run Tools->Boundary Polyline->Set Boundary Polyline. At the command line, there is a prompt:
Select boundary polyline:
Pick anywhere along the six sided perimeter polyline in the drawing.

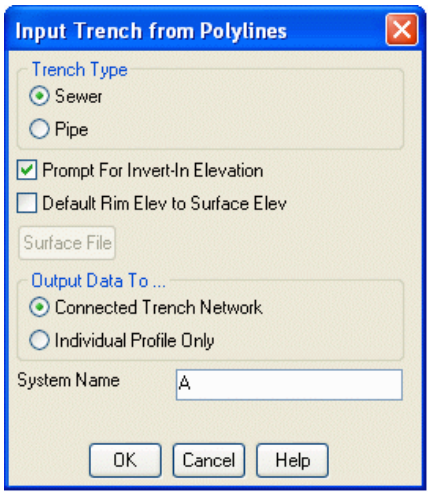
Now, to make the existing ground surface, run Tools->Make Existing Ground Surface. Then to make the design surface, run Tools->Make Design Surface.

Step 4 (Input Trench Network Data):

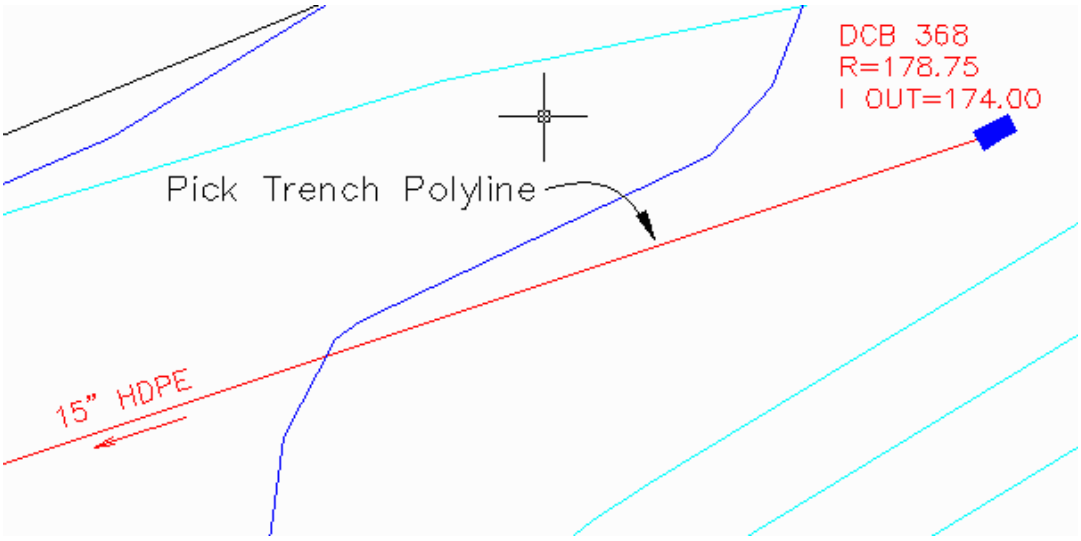
The trench network data consists of linked structures where each structure has a name, location (x,y), invert-in, invert-out and rim elevation. Each structure link has a pipe size. There are two ways of entering the trench data. When the drawing contains polylines for the trench lines and labels with the trench data, then you can use Input Trench From Polyline. Otherwise, there is the Create Trench Network Structure command which let's you pick the structure locations and enter the data in a dialog.

Method 1 (Input Trench Data From Polyline):
In this example, there is trench data already drawn in the drawing. Zoom in around the upper right area of trench line by running View->Zoom->Window and picking two corner points around this area.

Then run Trench->Input Trench From Polyline and an options dialog appears. In this case, we want Trench Type as Sewer because there are manhole rim elevations. Also Prompt For Invert-In Elevations is active since this example has a manhole with multiple connections with different invert-ins. And Connected Network is used so that the trench data can be used by the rest of the trench routines. The Individual Profile option will only create a profile (.pro) file. Fill out the dialog as shown and click OK.



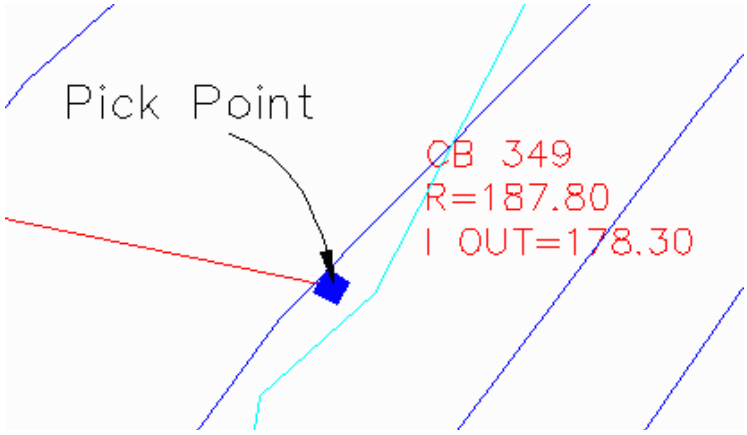
The rest of the prompting for this command is on the command line as the program walks through the trench line. For each point in the trench polyline, the program zooms the drawing to that point. The trench data can be picked from labels in the drawing. If the drawing doesn't have labels for the data, then you can enter the values.



Pick a polyline that represents a trench reach: *Pick the trench polyline*
Starting Station of trench reach <0.0>: *0.0*
For station 0.00 ...
Enter/<Select text of Manhole ID>: *Pick the DCB 368 label.* (If you had a drawing without a manhole ID label, then type E for Enter and enter the ID)
ID: DCB 368
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I Out=174 label.* (Since this is the upstream starting manhole, there really isn't a separate invert-in. So we are using the invert-out).
Invert-In: 174.000
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=174 label.*
Invert-Out: 174.000
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=178.75 label.*
Rim: 178.750
For station 201.44 ...
Enter/<Select text of Manhole ID>: *Pick the DCB 367 label.*
ID: DCB 367
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I In=172.85 label.*
Invert-In: 172.850
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=172.35 label.*
Invert-Out: 172.350
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=178.5 label.*
Rim: 178.500
Undo/Enter/<Select text of pipe size>: *Pick the 15" HDPE label.*
Pipe Size: 15.0
For station 327.09 ...
Enter/<Select text of Manhole ID>: *Pick the CB 347 label.*
ID: CB 347
Undo/Enter/<Select text of Invert-in elevation>: *Pick the I In=170.540 (CB 367) label.* (This is the invert-in for the connection to the CB 367 structure that this trench line connects to.)
Invert-In: 170.540
Undo/Enter/<Select text of Invert-out elevation>: *Pick the I Out=166.1 label.*
Invert-Out: 166.100
Undo/Enter/<Select text of manhole rim elevation>: *Pick the R=176.5 label.*
Rim: 176.500
Undo/Enter/<Select text of pipe size>: *Pick the 15" HDPE label.*
Pipe Size: 15.0
Another Polyline [<Yes>/No]? *N* for no.

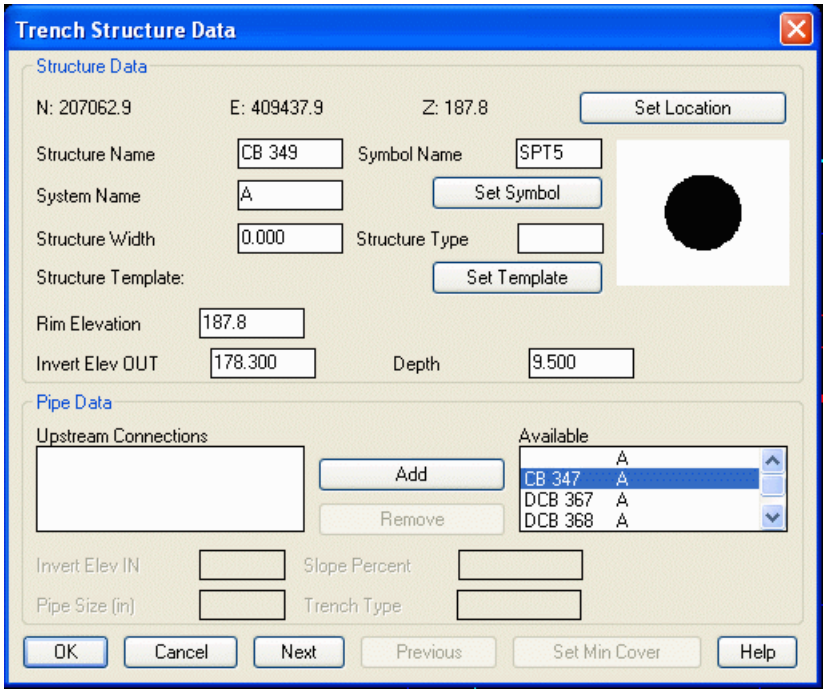
That completes this trench run and Takeoff draws its own trench polyline and labels.

Method 2 (Create Trench Network Structure):
The drawing contains another trench polyline and we could use Input Trench Data From Polyline again. Instead for practice, let's use the Create Trench Network Structure method. First we need to zoom to the new trench location. Run View->Zoom->Extents and then View->Zoom->Window and pick two points for a window around the lower right trench point (CB 349). Then run Trench->Create Trench Network Structure. At the command line, there is a prompt for how to locate the structure position. Choose Pick.
Locate by pick point, point number or station-offset [<Pick>/Number/CL]? *Pick*



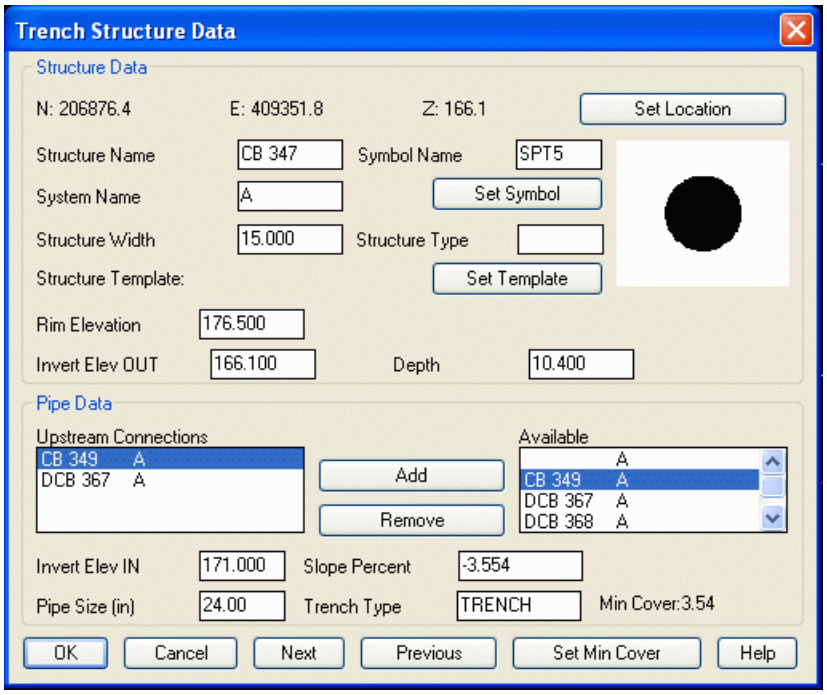
Next, there is a prompt to pick the position. To get the exact end point of the trench polyline, use the end point snap. The end point snap can be turned on by a number of different ways including the Settings->Object Snap command. In this case, type "end" and then space or enter. This puts the program in end point snap mode. Now move the pointer along the trench polyline until the end point snap icon is at the manhole location and then pick.
Pick structure location: *end of (pick point)*

Now a dialog appears for entering the structure data. Fill in the Structure Name as CB 349, the Rim Elevation as 187.8 and the Invert-Out as 178.3 and then pick OK.



All the structures are now created. The last step is to link this new structure to the network. We need to zoom to the next trench location. Run View->Zoom->Extents and then View->Zoom->Window and pick two points for a window around the left trench point (CB 347). Now run Trench->Edit Trench Network Structure and pick either the symbol for CB 347 or the label. Then a dialog appears with the data for CB 347. From the Available list, highlight CB 349 and pick Add. This creates a link from CB 347 to CB 349 and the link data is shown at the bottom of the

dialog. Enter the Invert-In as 171 and the Pipe Size as 24. Then pick OK.

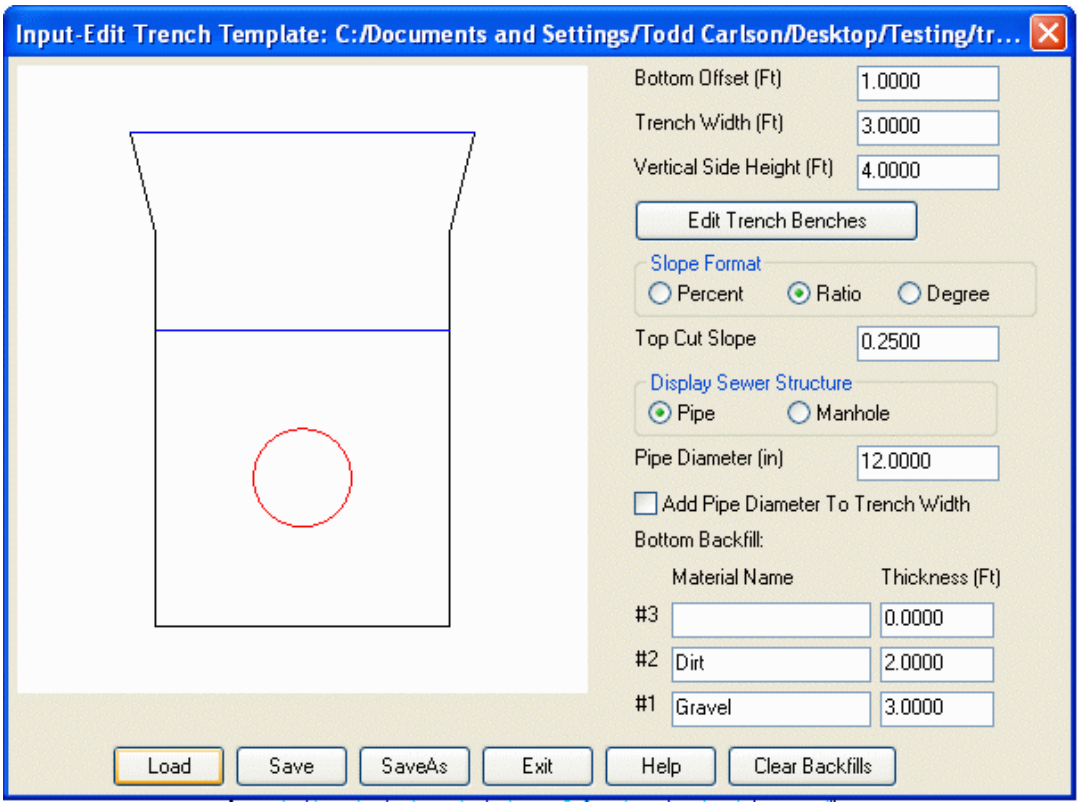


Step 5 (Input-Edit Trench Template):

The Trench Template defines the size of the trench for quantities. Run Trench->Input-Edit Trench Template. You are first prompted for a trench template file name. The Trench Template data is stored in a file that has a .tch extension. Choose the New tab, enter a file name like DEMO3 and then pick Open.

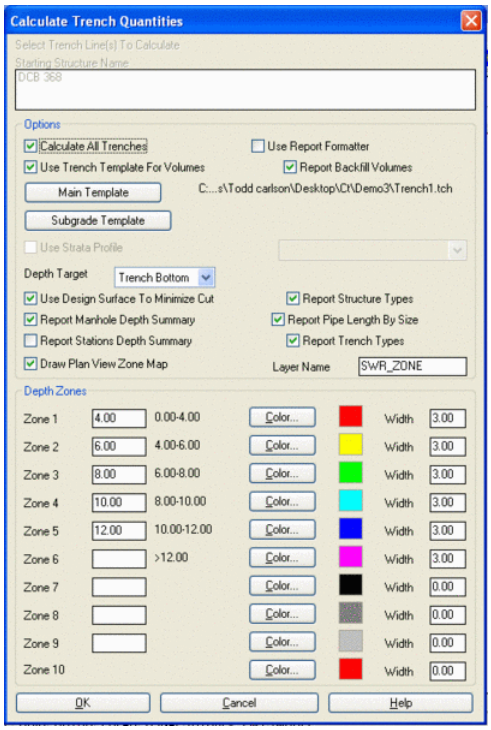
Next, there is a dialog for entering the trench dimensions. The Bottom Offset is the distance from the bottom of the pipe to the bottom of the trench. The Trench Width is the base width of the trench. The Vertical Side Height is the height from the bottom that the side walls are vertical until switching to the cut slope. If the surface is not reached by the vertical side height, then the cut slope is used for the rest of the distance to the surface. Edit Trench Benches allows you to set up to four benches in your trench. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. Note: This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. The Cut Slope can be entered in slope percent, ratio or degree format. The Backfill materials are optional. Up to three materials can be entered from the bottom.

Fill out the dialog as shown and pick Save and Exit.

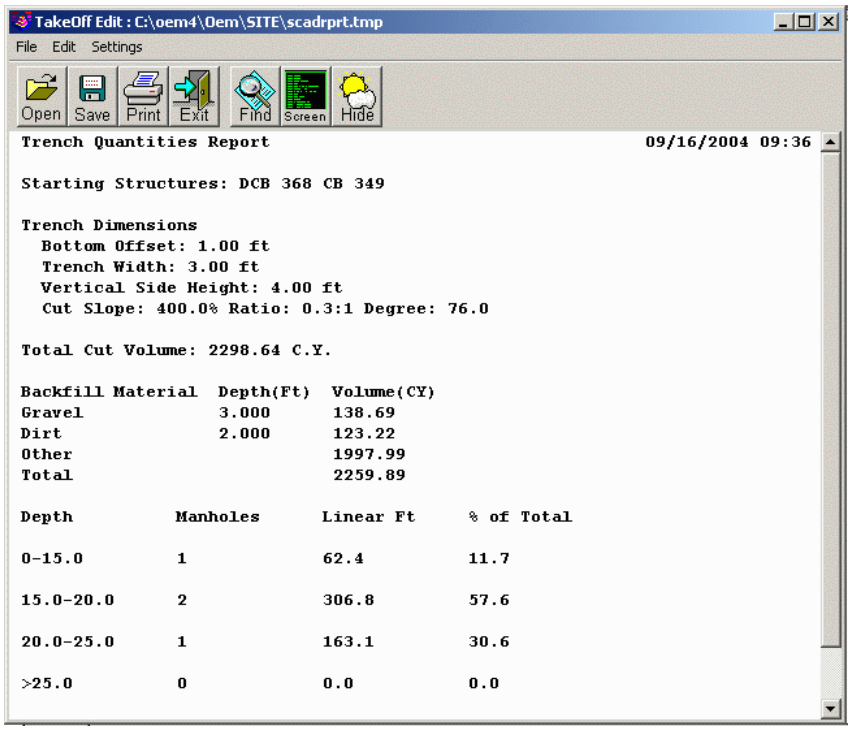


Step 6 (Trench Network Quantities):

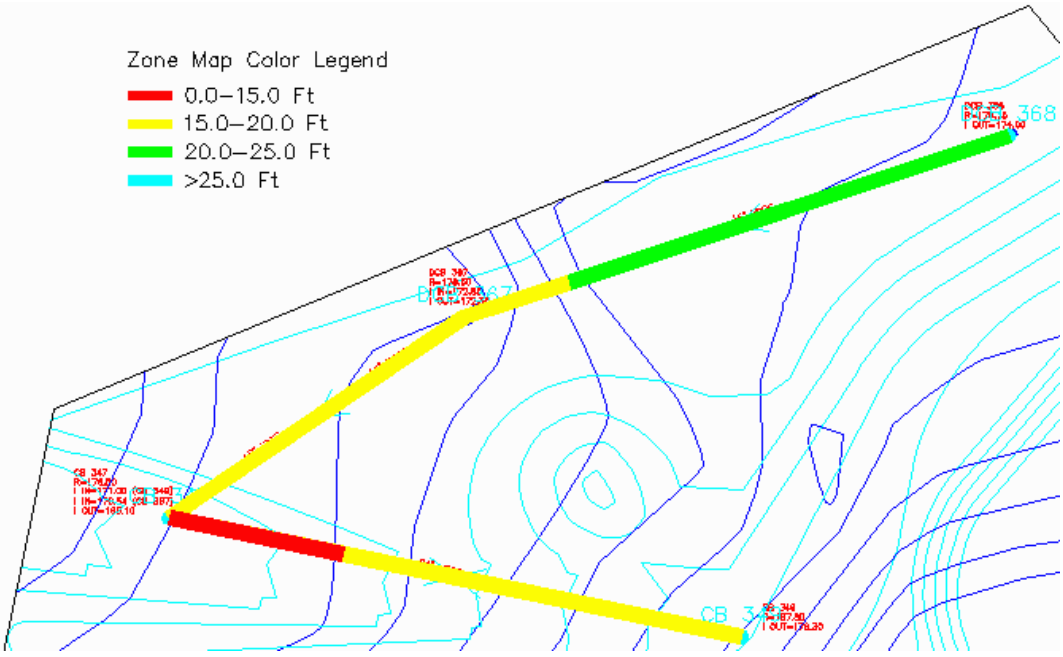
To calculate and report the trench quantities, run Trench->Trench Network Quantities. A dialog sets the report options. Check on Calculate All Trenches to get the quantities for the whole network. To get the trench cut volume, check Use Trench Template For Quantities and pick Set Trench Template and pick DEMO3.TCH. Also turn on Report Backfill Volumes to use our backfill material settings from the trench template. Finally, fill out the depth zones in the intervals that you are interested in. In this case, use 15, 20 and 25. The depth zones will be colored in the plain view. Once the dialog is filled out as shown, pick OK and the report is shown.



The report includes:
- The structure names at the start of each trench run included in the report.
- The trench template dimensions.
- The cut volume.
- The backfill volumes.
- The number of manholes and length of trench within each depth zone.

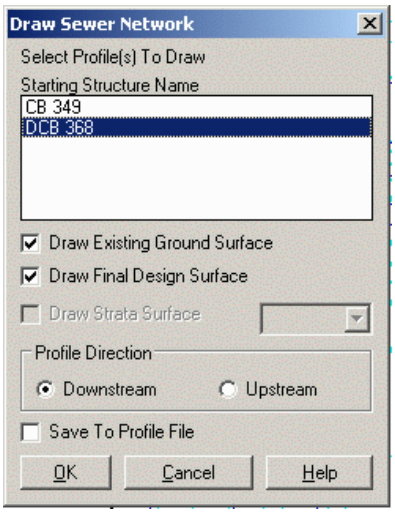

TakeOff Edit : C:\oem4\Oem\SITE\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Trench Quantities Report 09/16/2004 09:36
Starting Structures: DCB 368 CB 349
Trench Dimensions
Bottom Offset: 1.00 ft
Trench Width: 3.00 ft
Vertical Side Height: 4.00 ft
Cut Slope: 400.0% Ratio: 0.3:1 Degree: 76.0
Total Cut Volume: 2298.64 C.Y.
Backfill Material Depth(Ft) Volume(CY)
Gravel 3.000 138.69
Dirt 2.000 123.22
Other 1997.99
Total 2259.89
Depth Manholes Linear Ft % of Total
0-15.0 1 62.4 11.7
15.0-20.0 2 306.8 57.6
20.0-25.0 1 163.1 30.6
>25.0 0 0.0 0.0

The depth zones in the plain view with Zone Map Color Legend.

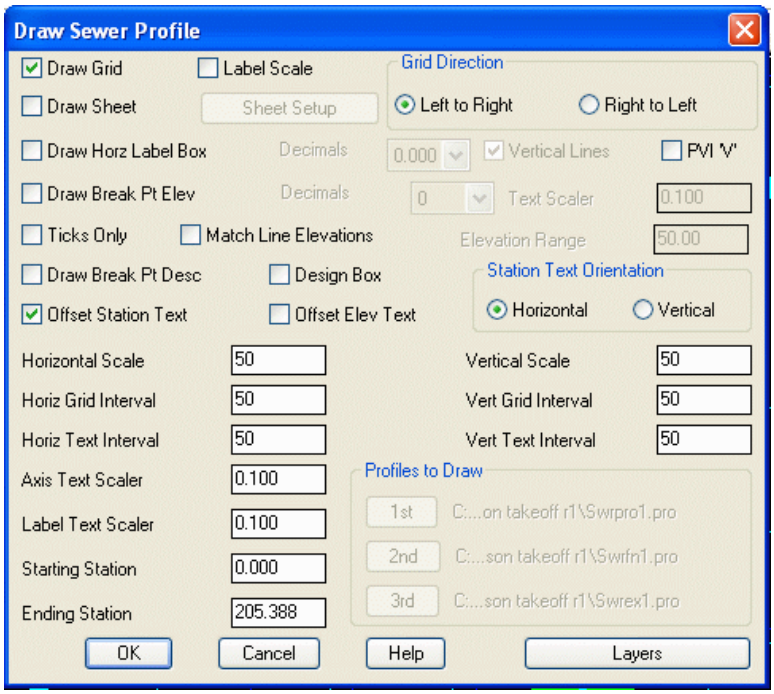


Step 8 (Draw Trench Network Profile):

To draw a profile of the trench line, run the Trench->Draw Trench Network (Profile) command. There is a dialog to select the starting structure for which trench line to process. Choose DCB 368. Also, there are options whether to draw the existing ground, design surface and strata surfaces if available. You can also choose the profile direction to go upstream or downstream. The Save To Profile File will create a profile (.pro) file for the trench. Fill out the dialog as shown and pick OK.

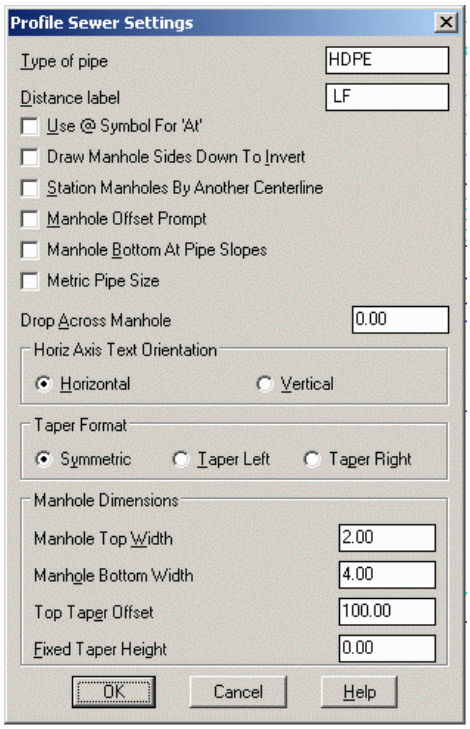


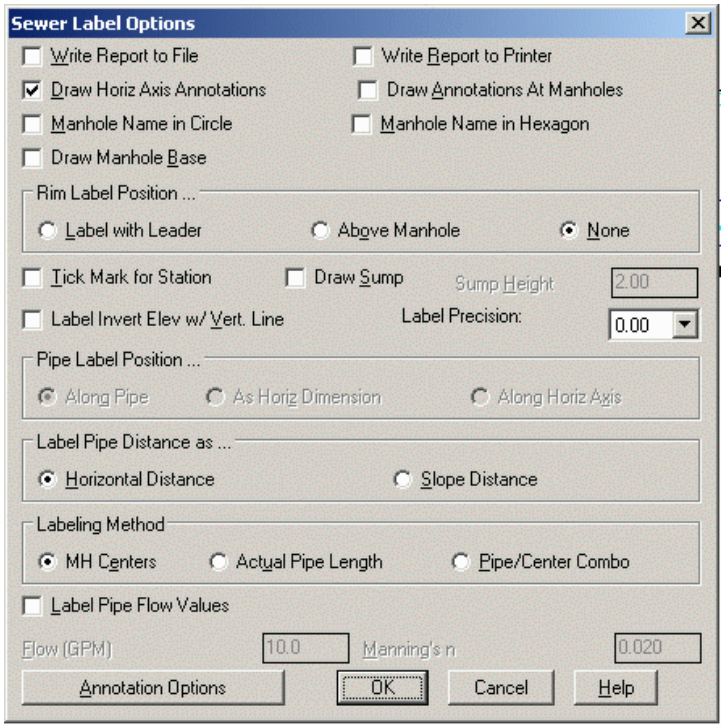
Next the Draw Profile dialog appears. Set the Horizontal Scale and intervals to 50 and the Vertical Scale and intervals to 25. This will make for two to one vertical exaggeration for the profile. When the dialog is filled out as shown, pick OK.



Next, there are prompts at the command line for the profile grid elevations and profile location.
Bottom Elevation of Profile Grid <150.0>: *Press Enter*
Top Elevation of Profile Grid <200.0>: *Press Enter*
Pick Starting Point for Grid <409458.0 , 207303.0>: *Pick a point in a blank area off to the side of the drawing.*

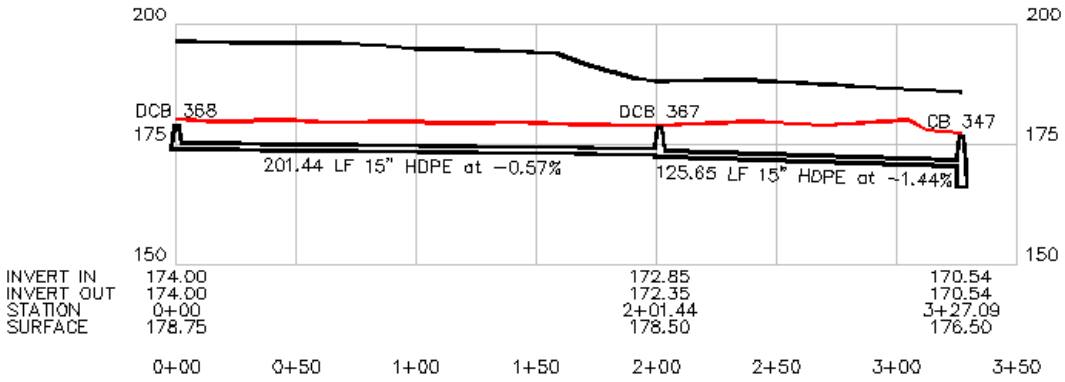
Next there are two dialogs with lots of settings for how to draw and label the trench profile. Let's go with the defaults except change the Type Of Pipe to HDPE. When the settings are ready, click the OK button for each dialog.





There is a final command line prompt for whether to use the manhole elevations. Enter Yes which will use the rim elevations defined in our trench network.
Use manhole elevations from profile [<Yes>/No]? *Y* for yes.

The profile shows the existing ground at the top, then the design surface and then the trench.



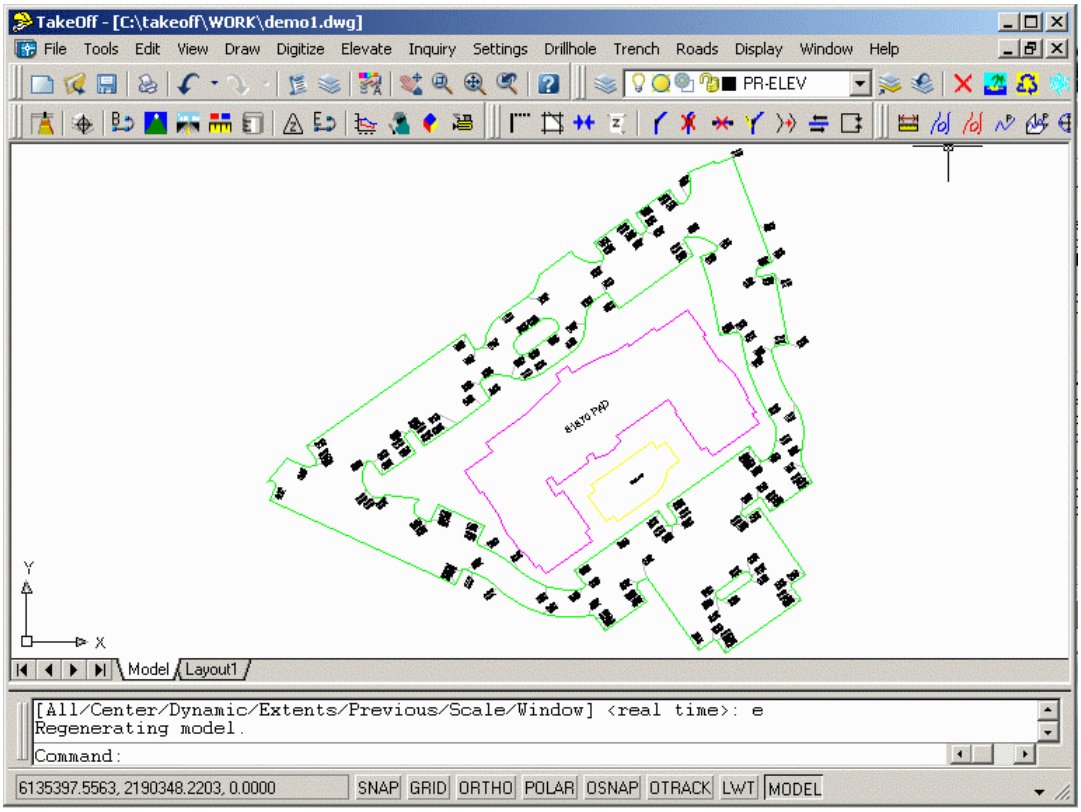
Drillhole and Strata

This lesson creates and processes drillhole data.

Step 1 (Run Lesson 1 Example):

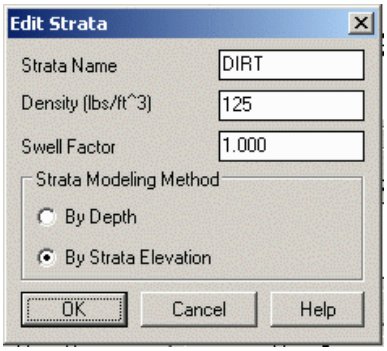
This drillhole lesson builds on the resulting drawing called demo1.dwg from the tutorial lesson 1 (CAD File Takeoff). Before continuing with this tutorial, run through and complete this lesson 1 tutorial.

When lesson 1 is done, let's set the display to show only the design entities. In the Display menu, turn off Existing Drawing and Other Drawing and turn on Design Drawing. Then run, View->Zoom->Extents. Now we're ready to add drillholes.


TakeOff - [C:\takeoff\WORK\demo1.dwg]
File Tools Edit View Draw Digitize Elevate Inquiry Settings Drillhole Trench Roads Display Window Help
PR-ELEV
Model Layout1
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: e
Regenerating model.
Command:
6135397.5563, 2190348.2203, 0.0000
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL

Step 2 (Drillhole/Strata Settings):

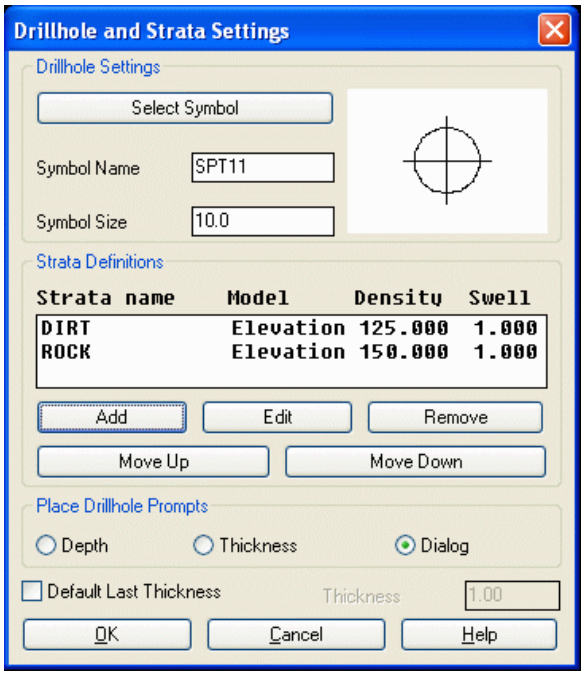
From the File menu, choose Drillhole/Strata Settings. This command sets the drillhole symbol and the default strata names. For this tutorial, we are interested in rock quantities and we need to define two strata: Dirt (material above the rock) and Rock.



Pick the Add button which brings up another dialog that defines a strata. Enter a strata name of "DIRT" and a density of 125 which will be used to calculate tons in the volume report. You can also have a strata specific cut swell factor. The strata can be modeled either by the elevations from the drillholes or by the depth from the existing ground. In this case, we will model by strata elevation. When the dialog is filled out as shown, pick OK.

Next, pick the Add button again. This time, fill out the dialog with a strata name of "ROCK" and density of 150. Then pick OK.

The Strata Definitions in the main dialog need to be in top to bottom order. To change the order, highlight a strata name and use the Move Up or Move Down buttons. In this case, we want Dirt then Rock. Click OK now from the main dialog.



Step 3 (Input Drillhole Data):

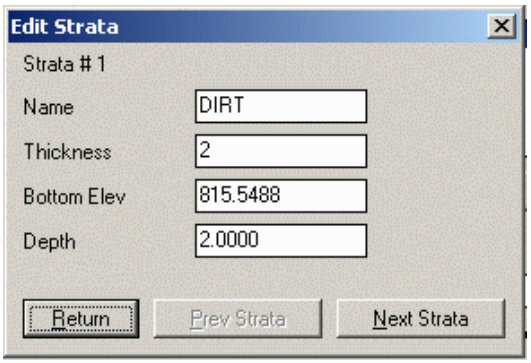
There are two different methods for entering drillhole data into Takeoff: Drillhole Import and Place Drillhole. Drillhole Import reads the drillhole data from a text file. This command supports customizing the sequence of drillhole data fields to match the format of the text file. Place Drillhole creates the drillholes at picked positions in the drawing and enters the data in a dialog. For this tutorial, we will use Place Drillhole.

Run the Drillhole->Place Drillhole command. At the command line, there is a prompt to pick the drillhole location. If you know the coordinates for the drillhole, you can type in the easting,northing instead of picking on the screen. In this case, let's pick a point above the upper right of the main building.
Pick Drillhole Location: *pick a point*



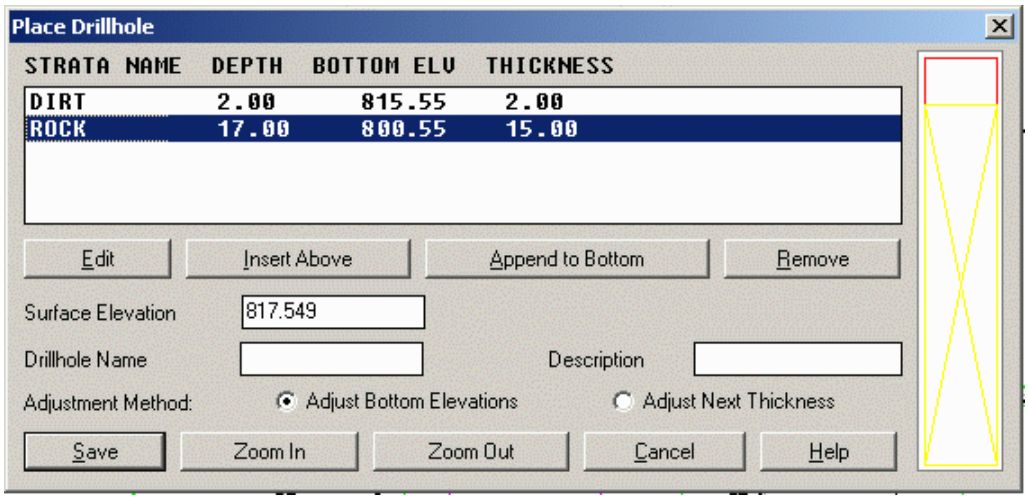
Then there is a dialog for entering the drillhole data. The surface elevation is automatically filled in using the existing ground surface model. The Drillhole Name and Description are optional. The list of strata defaults to the strata defined in Drillhole/Strata Settings. Each strata defaults to a thickness of zero. To set the strata thickness, highlight the strata and pick the Edit button.

For this case, highlight Dirt and pick Edit. This brings up the Edit Strata dialog. The strata position can be defined by thickness, elevation or depth. Setting any one of these fields will update the other fields. For our dirt stata, fill in a thickness of 2 and then pick OK.

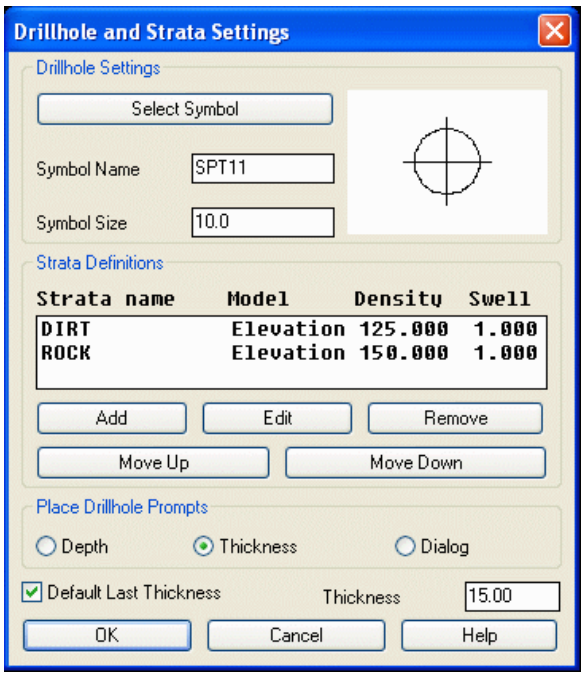


Next, pick Rock from the strata list and pick Edit. For this example, we only know the depth to the top of rock depth and not the total rock thickness. We will treat all cut below the top of rock as rock strata. So we will set the rock thickness deep enough to be lower than the deepest cut on site. In this case, we will use a rock thickness of 15. So in the Edit Strata dialog for rock, enter a thickness of 15 and then pick OK.

After editing the rock strata, we are returned to the main Edit Drillhole dialog. Pick the Save button.



Now let's locate two more drillholes using a different method. Return Drillhole/Strata Settings dialog and change Place Drillhole Prompts to Thickness. Also, check on Default Thickness and set it to 15 feet, Press OK.



Now run Place Drillhole again and for the second drillhole, pick a position in the lower parking lot. The command line will prompt you to enter a dirt thickness, type in 1.5 and your drillhole is created. For the third drillhole, pick a position left of the main building. Enter a dirt thickness of 3.0, save, and enter to end the command.

Step 4 (Make Strata Surfaces):

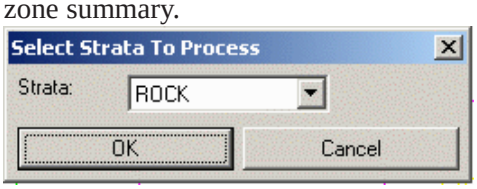
Now that the drillholes are in the drawing, to make the strata triangulation surfaces, run the Drillhole->Make Strata Surfaces command. There are no prompts for this routine. The strata surfaces are modeled from the drillholes and saved with the project. The file names for the strata surfaces use the drawing name plus "-ch#" where the # is the strata sequence number. For this example, the file names will be "demo1-ch1" for bottom of dirt and "demo1-ch2" for bottom of rock.

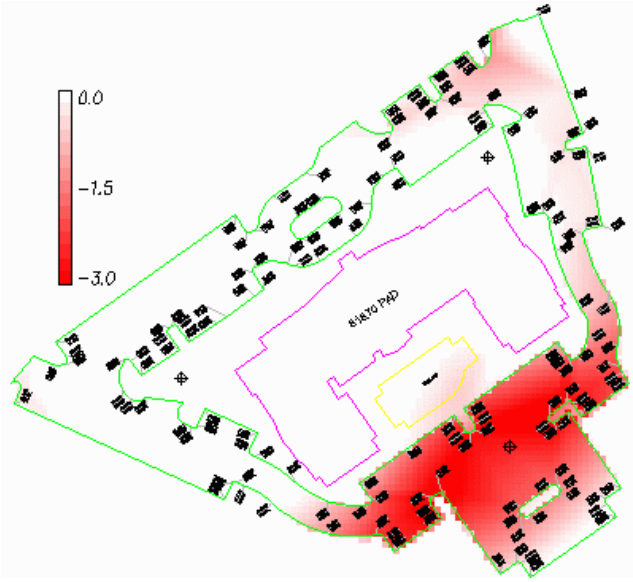
Now that the strata surfaces are created, there are several Takeoff routines that will use these surfaces such as:
- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- Cut/Fill Labels
- Surface Inspector
- Quick Profile
- Trench Network Quantities

Step 5 (Draw Strata Cut Color Map):

From the Drillhole menu, pick Draw Strata Cut Color Map. This command compares the design surface with the strata surface to make a cut color map of the cut depths for the strata. This command is one way to verify that the strata surfaces are modeled correctly.

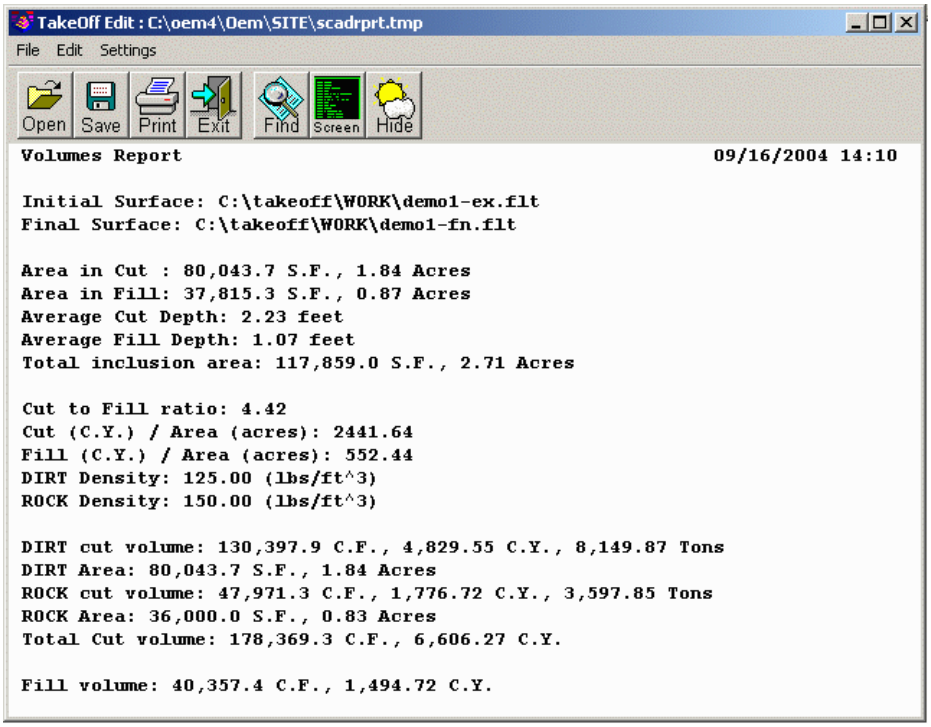
There is a dialog to select which strata map to draw. Choose Rock and pick OK. Then there is an option to draw a cut depth legend. Pick a position for the legend in the upper left of the site and use the defaults for size and zone summary.





Step 6 (Calculate Total Volumes):

Run the Tools->Calculate Total Volumes command. When strata surfaces are defined, the volume routine will breakout the cut volume into the different strata. The resulting dirt and rock quantities are shown in the report.


TakeOff Edit : C:\oem4\Oem\SITE\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 09/16/2004 14:10
Initial Surface: C:\takeoff\WORK\demo1-ex.flt
Final Surface: C:\takeoff\WORK\demo1-fn.flt
Area in Cut : 80,043.7 S.F., 1.84 Acres
Area in Fill: 37,815.3 S.F., 0.87 Acres
Average Cut Depth: 2.23 feet
Average Fill Depth: 1.07 feet
Total inclusion area: 117,859.0 S.F., 2.71 Acres
Cut to Fill ratio: 4.42
Cut (C.Y.) / Area (acres): 2441.64
Fill (C.Y.) / Area (acres): 552.44
DIRT Density: 125.00 (lbs/ft^3)
ROCK Density: 150.00 (lbs/ft^3)
DIRT cut volume: 130,397.9 C.F., 4,829.55 C.Y., 8,149.87 Tons
DIRT Area: 80,043.7 S.F., 1.84 Acres
ROCK cut volume: 47,971.3 C.F., 1,776.72 C.Y., 3,597.85 Tons
ROCK Area: 36,000.0 S.F., 0.83 Acres
Total Cut volume: 178,369.3 C.F., 6,606.27 C.Y.
Fill volume: 40,357.4 C.F., 1,494.72 C.Y.

Digitizing

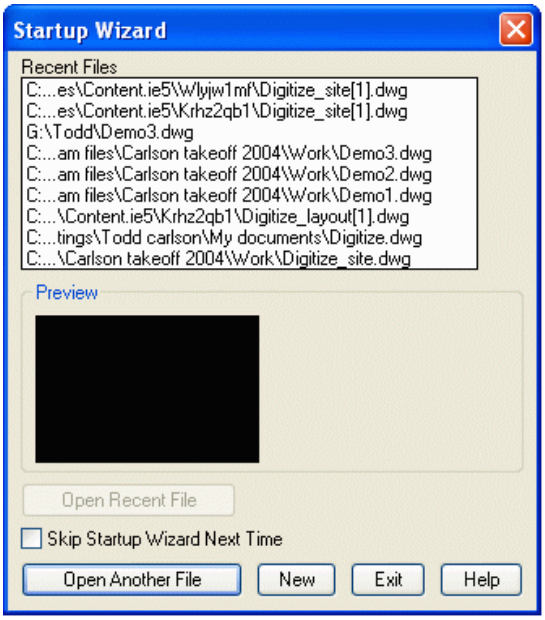
This lesson tranfers a paper plan into Carlson Takeoff.

Step 1 (Setup):

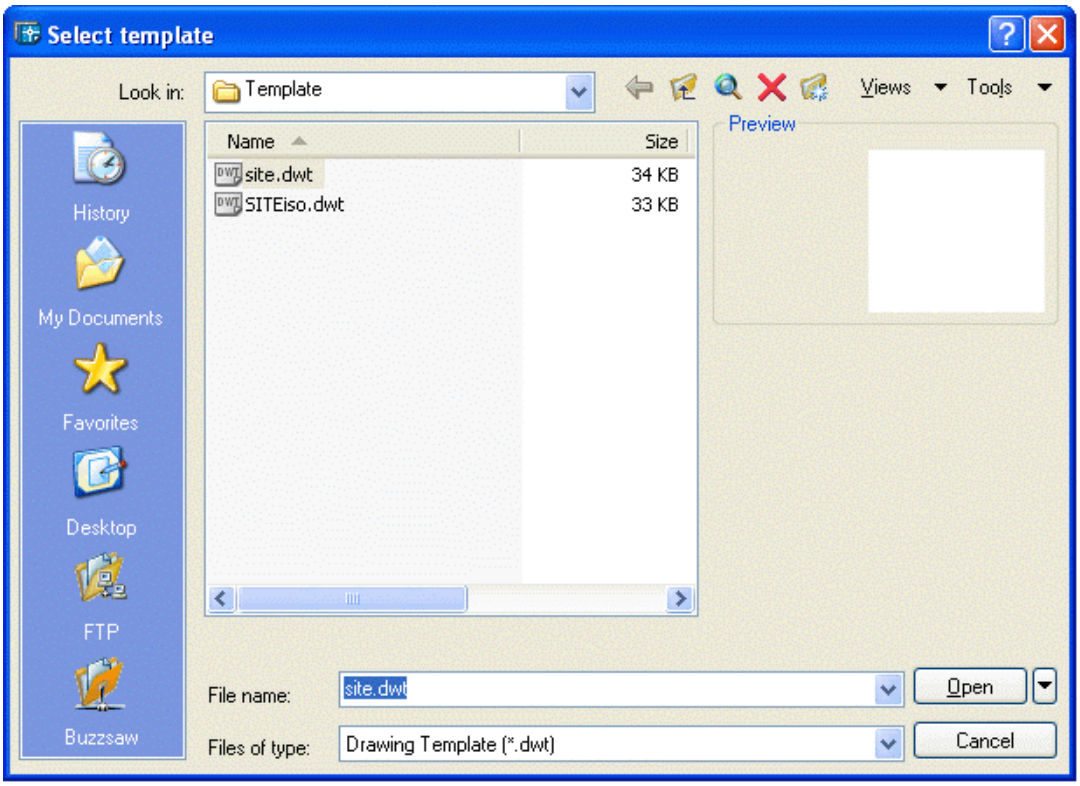
To digitize in Carlson Takeoff, you need to install the Wintab digitizer driver. See Digitizer Setup in the manual if you have not installed or have problems with the Wintab driver. If Wintab is installed, then make sure your drawing board is on and take the paper plan provided with the Carlson Takeoff manual and place it on your drawing board.



Now, click on the Windows icon for Takeoff to launch the program. You may be presented with a "Startup Wizard" dialog similar to the one shown below, if so click New.



If a Startup Wizard did not appear, then under File menu, select New to start a new drawing. You will be prompted for a template to use. Templates determine the default settings for your drawing. For this tutorial, select site.dwt and click Open.

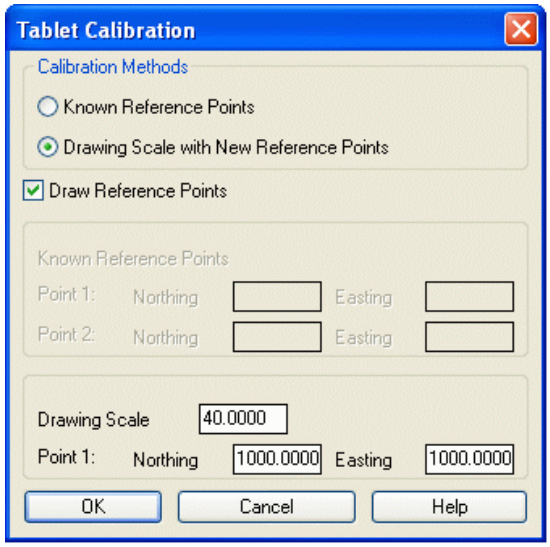


Next, the New Drawing Wizard appears for setting the drawing name. Click on the Set button at the top dialog. In the file selection dialog, enter the file name of "digitize" and pick the Save button. Then Exit the New Drawing Wizard. From here, a Data Files dialog appears where no changes are needed. Pick the Exit button.



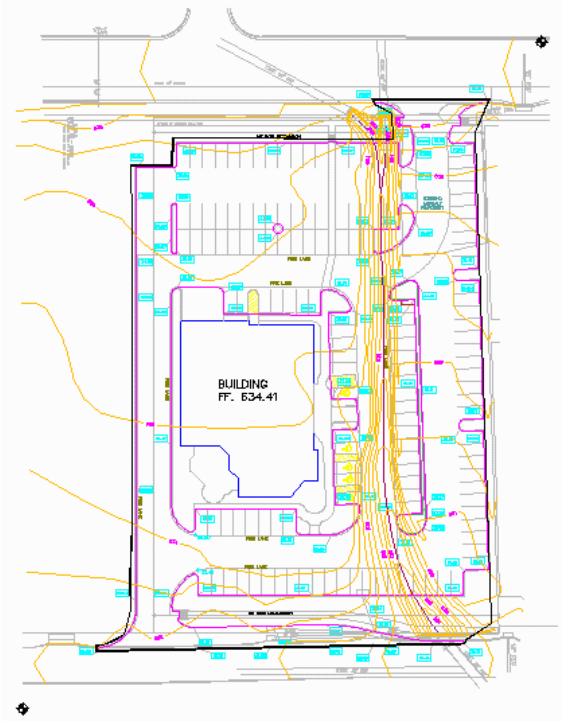
Step 2 (Tablet Calibration):

To start things off, you need to set the coordinate system for the paper plan by running the Calibrate command under Digitize menu and sub-menu Tablet. Calibration is required to let the program know the orientation and scale of the paper plan.



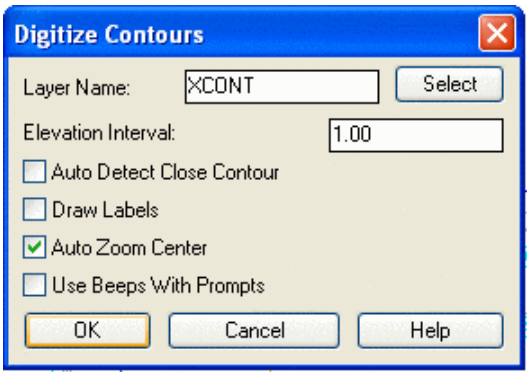
There are two different Calibration Methods: Known Reference Points and Drawing Scale with New Reference Points. Known Reference Points allows you to enter in the coordinates of two marked points on the paper plan. This method applies when you know the coordinates of at least two points on the paper plans. Drawing Scale with New Reference Points allows you to setup a coordinate system for the plans by entering the plan scale and picking any two points from the paper plan with the digitizer puck.

In this case, we will use Drawing Scale with New Reference Points. First, enter in the Drawing Scale listed on the paper plan. On this drawing, the scale is 1:40, so enter in 40. Use the default coordinates for Point 1 and click OK. Now Carlson Takeoff will prompt you for your First and Second Reference points. Generally, you want to pick to points on the drawing that you can find and use again in case you need to recalibrate. Also, the further away the points are from each other, the more accurate the coordinate system will be. With the digitizer puck, pick on the ⌖ icon in the lower left and upper right of the drawing for the two Reference Points. The firstpoint is assigned the coordinates of 1000,1000 from the dialog and the second point is assigned coordinates to match with the plan scale. From now on, all of your points will be in relation to these two points.



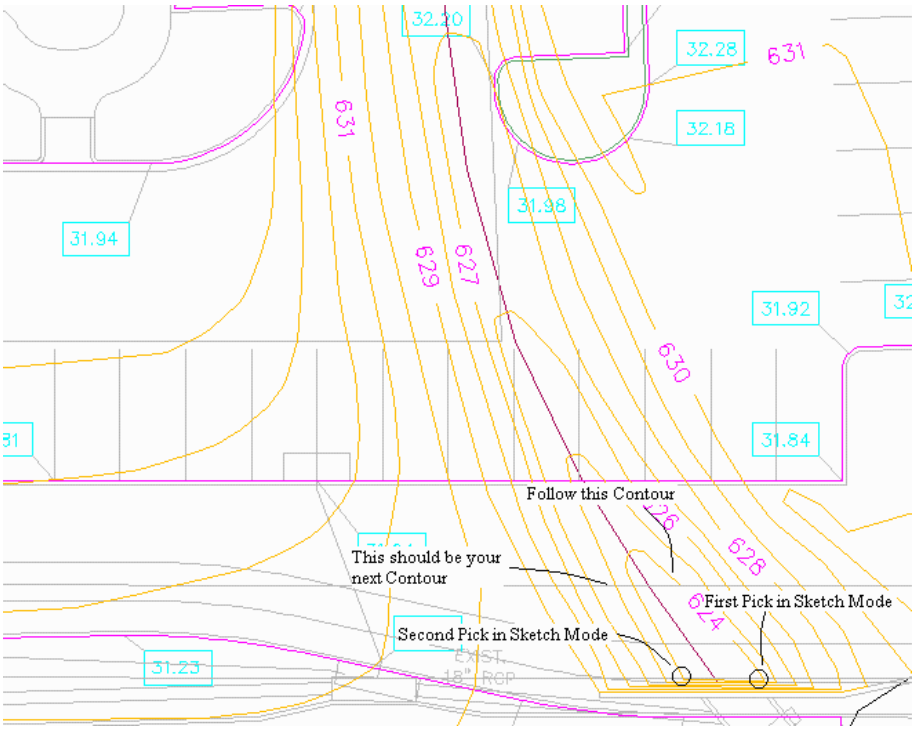
Step 3 (Digitizing Existing Contours):

We will now digitize the existing contours. Under the Digitize menu, click on Existing and then go to Contour Polyline, and this dialog will appear. Enter in a Layer Name of XCONT and select OK. Note: your Elevation Interval should match the intervals marked on your paper drawing. In this drawing, the interval is the same as the default of 1.00.



The rest of the prompting occurs at the command line and starts with the contour elevation. Find the lowest elevation for the existing contours labeled in bottom right corner of the paper plan zoomed in on below. In this example, the lowest elevation is 624 feet. The elevation can be entered either with the digitizer puck keys or with the computer keyboard. The layout of the digitizer keys is set in Digitizing Settings->Puck Layout. Press Enter

after you have entered in 624. You want to enter in the lowest contour so that as Carlson Takeoff adds the Elevation Interval, it is from lowest to highest.



Next, you will see the following prompt:

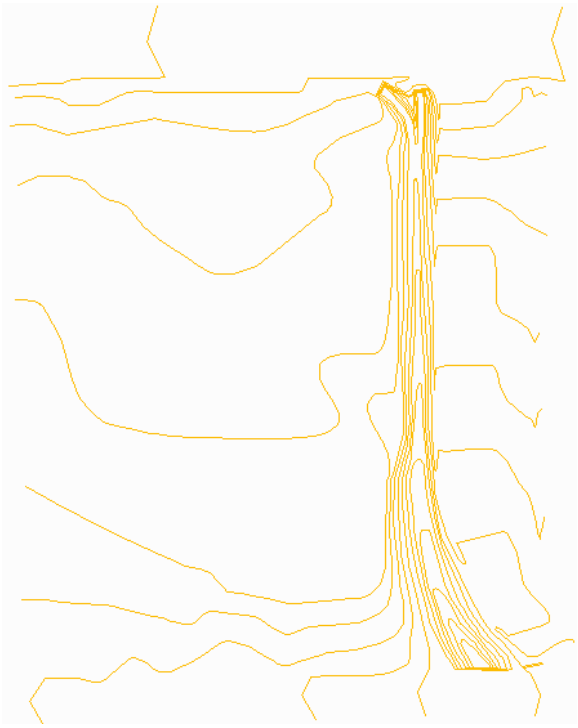
Sketch[0]/Exit[A]/Pick the first point:
There are two different ways to digitize: in Pick Mode or Sketch Mode. You can switch between them at anytime. In this tutorial we will run through how to do both. For now, type in [0] and press enter to get into Sketch Mode. In Sketch Mode, you will be prompted to Pick and drag. The point you pick is the starting point of a contour. Drag is asking you to follow that contour with the digitizer puck on the paper plan. Click a second time when you have traced the entire contour and have reached the end of the contour. You will then be prompted as follows:

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end):

Type in [B] for Undo if you made a mistake and need to sketch part of the contour again. [A] will close the contour, and [0] will switch you into Pick Mode. We still have more existing contours to digitize, so press Enter to end and answer yes to the Digitize Another Contour prompt. Takeoff will prompt you to verify the elevation. Remember, we set the Elevation Interval to one, so the default elevation for your next contour line is 625, press Enter. Now, pick the endpoint of the next contour and trace it in the same manner as the previous contour.

Now let's try Pick Mode. Say yes to digitize another contour and check to see if the default elevation corresponds with the contour your about to digitize. If not, simply type in the correct number in the command line. Next, pick [0] to get into Pick Mode. In Pick Mode, you do not have to trace the contour. Rather, pick with the digitizer puck to create points that will make up the contour. Note: Less picks are needed on fairly straight segments.

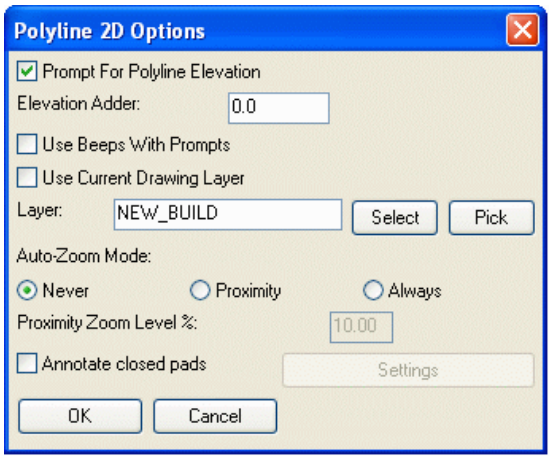
Conversely, more picks will give you a more acurrate contour. Press Enter when you have reach the end of the contour. Repeat this until you have digitized all of the existing contours you want to have in Takeoff (see below).



Step 4 (Digitizing the Design):

Now we will digitize the building and curb linework of the Design Surface using the Digitize 2D Polyline and 3D Polyline commands. Besides drawing the linework positions, we will also assign layer names to the linework that we will use later to identify the types of linework. In this example, there are no design contours, only the design building and curb linework and spot elevations.

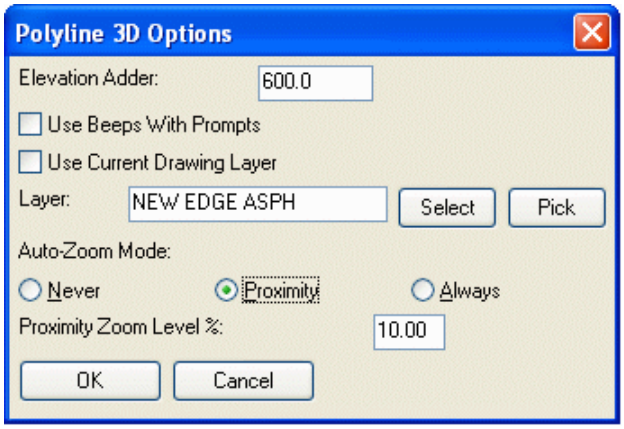
Let's begin by digitizing the main building. Under Digitize, check on Design and go to 2D Polyline. 2D Polyline is used to digitize linework enities with one elevation. Toggle off the check box **Use current drawing layer** and name the layer NEW BUILD. Toggle on the **Prompt For Polylne Elevation** option. Then click OK.



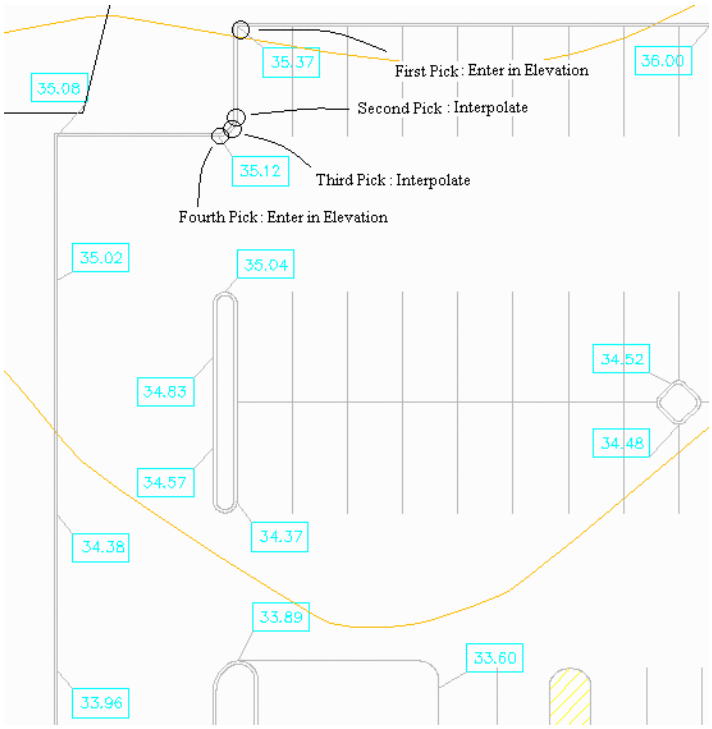
At the command line, enter in the building elevation of 634.41 found labeled in the middle of the building and press Enter. Then pick the points that define the building outline. Start in the upper left corner and pick at every corner around the building. When you have picked around the entire building, type in [A] for close to finish digitizing the building.

Enter polyline elevation <0.00>: *634.41*
First point: *pick a building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the next building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the next building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick the last building point*
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *A to close*
Digitize Another NEW_BUILD Polyline [Yes(A)/<No(B)>]? *B for No*

Notice that the parking lot linework consists of different elevation levels. To digitize entities with more than one elevation, go to Digitize and select 3D Polyline from the pull-down menu. Make sure that the **Prompt For Polyline Elevation** option is on, the **Use current drawing layer** toggle is off and name the layer NEW EDGE ASPH.



Let's start by digitizing the parking lot starting from the zoomed in section below. The edge of asphalt is the inside line. The parking lot elevation labels have been shortened on the paper plan. For example, they read 35.37 and 35.12, when the actual elevations are 635.37 and 635.12. Enter in 600 as the Elevation Adder, then click OK.



Click on the point with the digitizer puck where the 35.37 elevation label points to in the upper left corner of the parking lot. When prompted for Elevation enter in 35.37. Pick below the first point where the linework starts to curve. We do not have an elevation for this point, but we can interpolate the elevation from the two points around it using the interpolate option. Type in I for interpolate or hit the A button on the Puck. Next pick the middle point of the curve and again use Interpolate for the elevation. Next pick the end of the curve at the 35.12 label and enter in the elevation 35.12. Continue digitizing for the rest of the edge of asphalt linework. Digitize each point where there is an elevation label and each point where the curb line changes direction.

The first prompts should resemble these:

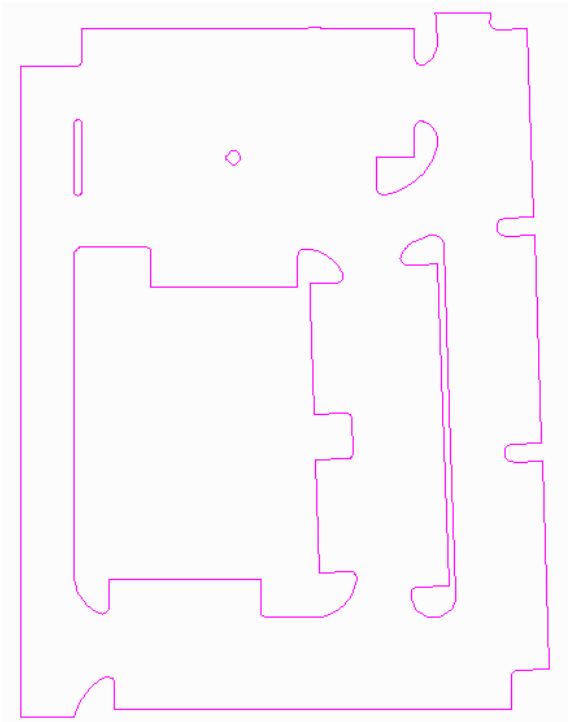
First point: *pick first point (at 35.37 label)*
Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: *35.37*
Z: 635.37
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): *pick next point (start of curve)*
Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <635.37>: Press the [A] button on the Puck for Interpolate

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *pick next point (middle of curve)*
This point elevation will be interpolated upon completion.
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *pick next point (end of curve, at 35.12 label)*
This point elevation will be interpolated upon completion.

Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: *35.12 (Enter)*

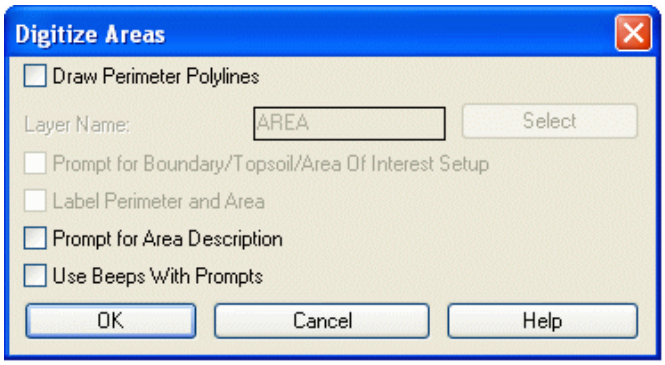
To check the elevations of the interpolated points go to List under the Inquiry menu and click on the poly-line you just created and press Enter. A text window will appear showing you the layer name, coordinates, and elevation of each point. To return to the main graphic screen, press F2.

Use the 3D Polyline command to digitize the rest of the parking lot as seen below.



Step 5 (Area):

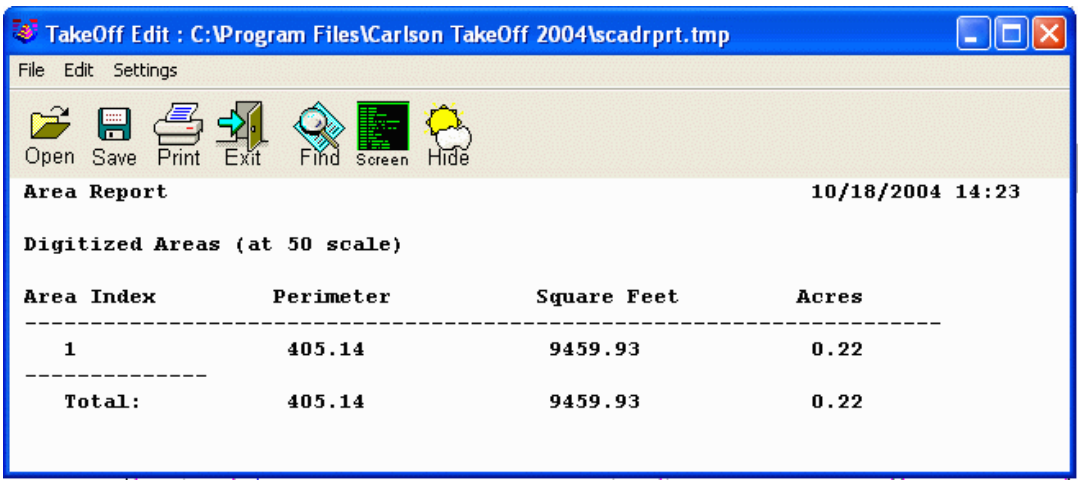
Now that we have digitized the Design Surface, let's check the Area of certain sections. Select Area under the Digitize Menu and match the below dialog.



To approximate the area of the main building, pick the points of the building outline.

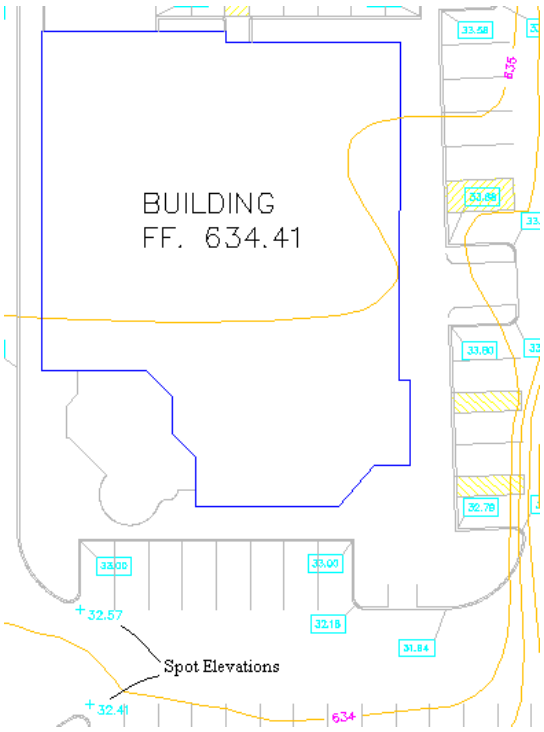
Command: *dig_area*
Pick starting point: *Pick points as close to the building design linework as you can*
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):
Undo[B]/Pick next point (Enter to end):

Digitize Another Area [<Yes(A)>/No(B)]? *B*
When finished with the building points, press Enter to end. Then answer no for no more areas. Takeoff will then display an Area report similar to the one shown below.

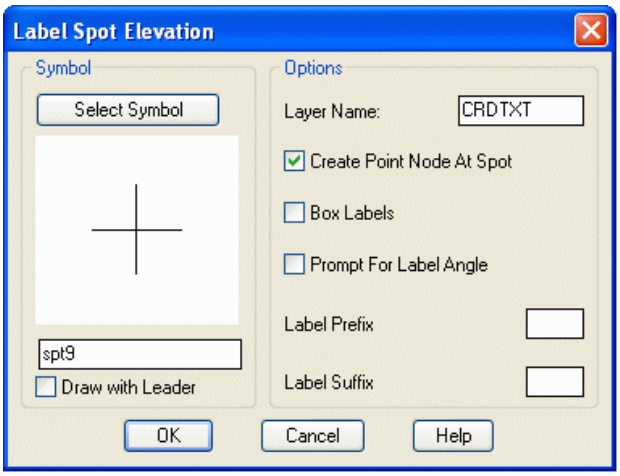

TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Area Report 10/18/2004 14:23
Digitized Areas (at 50 scale)
Area Index Perimeter Square Feet Acres
1 405.14 9459.93 0.22
Total: 405.14 9459.93 0.22

Step 6 (Spot Elevations):

In our paper drawing, we have two spot elevations labeled 32.57 and 32.41 shown in the bottom left below.



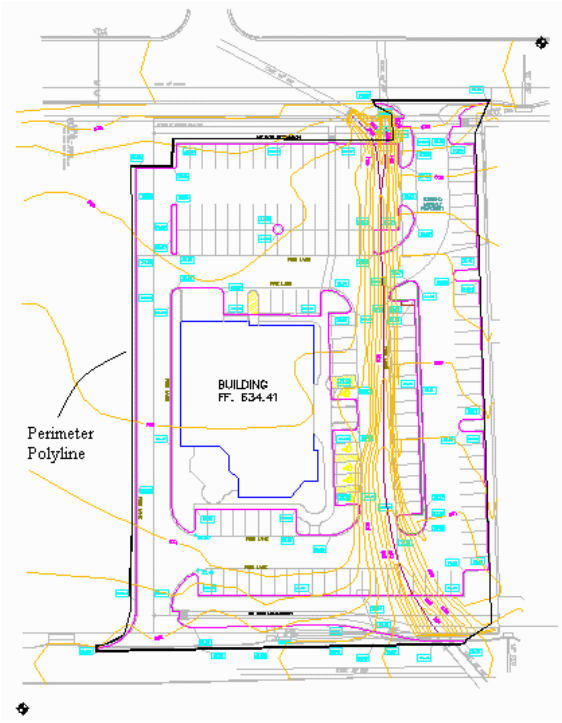
To digitize these elevations, we can use the Spot Elevation command under the Digitize menu. Fill out the Spot Elevation dialog as shown and pick OK.



In the paper plan, find and click on the spot elevations with the puck. When prompted, enter in their corresponding elevations of 632.57 and 632.41.

Step 7 (Boundary Polyline):

The limits of the site are defined by a closed polyline. This polyline is used as the boundary for the models and the volumes. Under the digitize menu, check on Other and then select Perimeter. Type in PERIMETER as the layer name. Now digitize around the bold, outside line shown below.



Say No to the prompt: **Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]?**

Now run Tools->Boundary Polyline->Set Boundary Polyline and pick the perimeter polyline. This selected polyline is now set as the boundary polyline for the rest of the Takeoff routines.

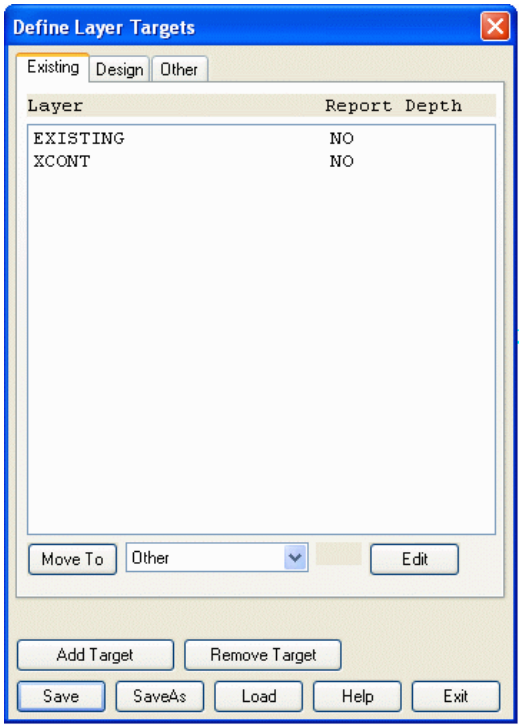
Step 8 (Layer Targets):

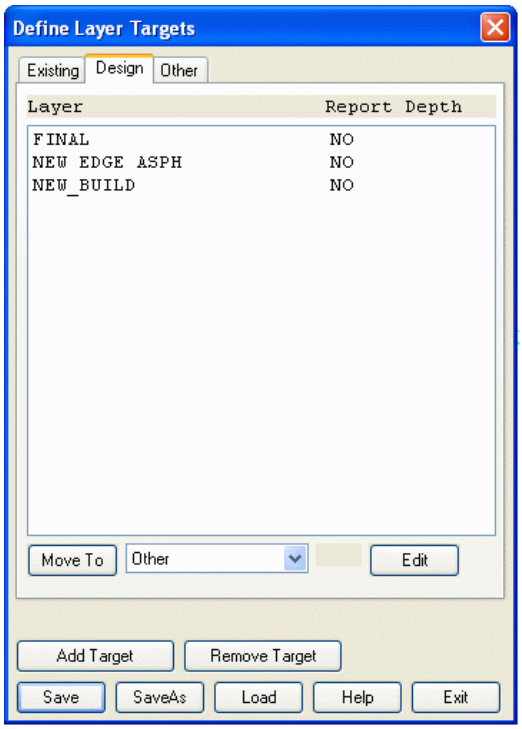
From the Tools menu, choose Define Layer Target/Material/Subgrade. Every entity (line, polyline, point, etc) in the drawing is assigned a layer name. Takeoff uses the entity layer names to define which entities are for the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces. The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours

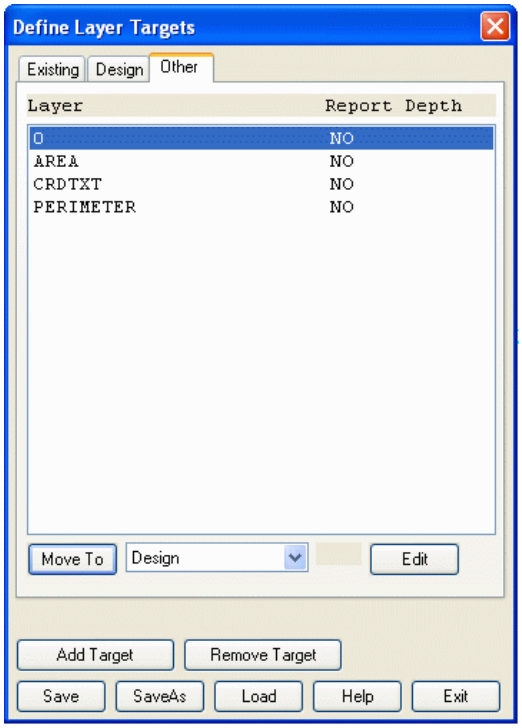
are on the layer "NEW", then "NEW" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other.

The Define Layer Targets dialog has three lists of layers: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. We have already defined the layers for their correct targets. We did this by check on Existing, Design, or Other in the pull-down menu.

Check that your Layer Targets resemble the three lists shown here. If a layer is out of place, highlight it, and hit the "Move To" button after selecting the correct target to send it to. After reviewing, pick Save and Exit.







Now that the layer targets are defined, there are several commands that can be applied. In the Display menu, you can turn on/off whether to display layer targets by using Existing Drawing, Design Drawing and Other Drawing, or by right-clicking with your mouse. For example, when Design Drawing is checked, then picking this menu item will uncheck it and turn off all the layers for the design surface. Likewise, picking Design Drawing when it is unchecked will make it checked and turn on the design surface layers.

Practice turning on/off the Existing, Design and Other Drawing in the Display menu. When only Existing Drawing is on, you should see just the contours. When only Design Drawing is on, you should see just the design polylines

and leader labels. When only Other Drawing is on, you should see the entities that are assigned to neither existing nor design.

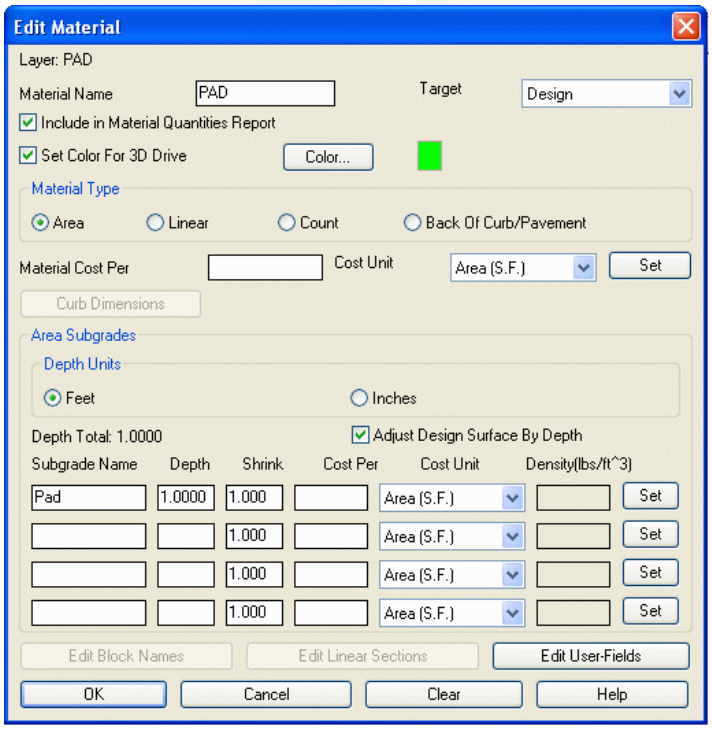


Step 9 (Define Material/SubGrade):

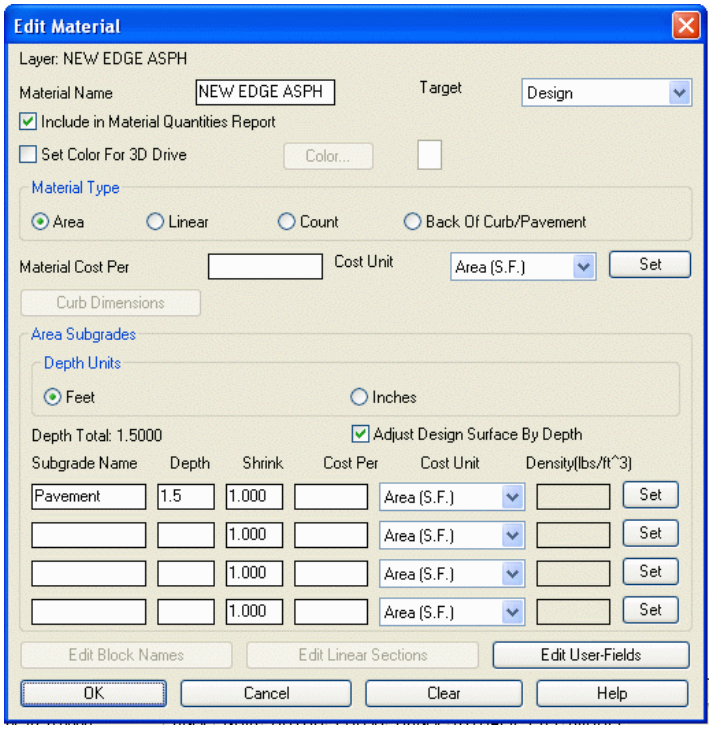
Besides assigning target surfaces by layer, layers are also used to define material names and subgrade depths. By assigning material names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. The polylines used for subgrade depth must be closed polylines. Takeoff supports nested subgrade polylines for exclusion areas such as islands by counting how many subgrade polylines surround an area. If the number is odd, then the area is inside the subgrade. Otherwise the area is not part of the subgrade.

First, let's confirm the layer names for our subgrades. Go to the Display menu and check on Design Drawing, uncheck Existing Drawing and uncheck Other Drawing. Then run Inquiry->Layer ID and pick the large pad polyline. It reports that this layer is NEW BUILD. Next use Layer ID to pick the curb polyline. It reports that this layer is NEW EDGE ASPH.

Now run Define Layer Target/Material/Subgrade and pick the Design tab. Highlight layer NEW BUILD and pick the Edit button. A dialog appears for defining the pad material properties. Check on the Include In Material Report option, enter the Material Name as "Pad", set the first subgrade name to "Pad", and set the Depth as 1. Once the dialog is filled out as shown, pick OK.

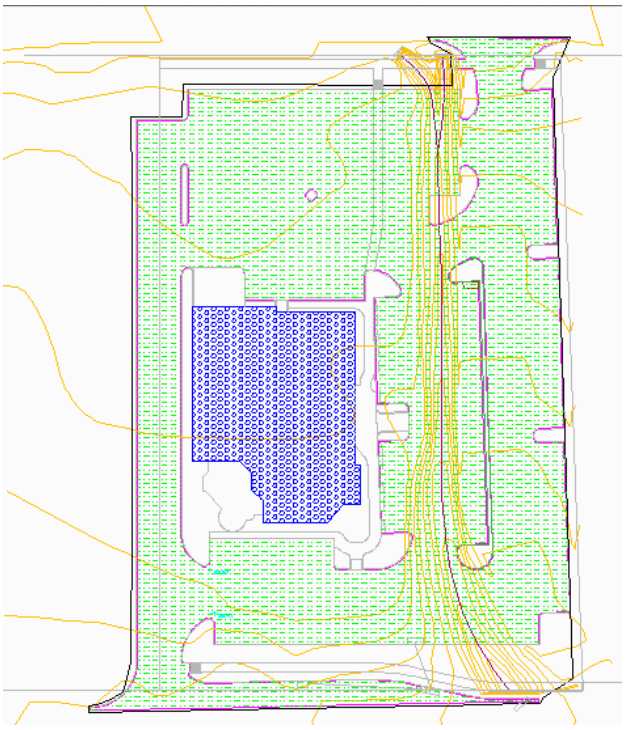


Next pick layer NEW EDGE ASPH and choose Edit. In the Edit Materials dialog, check on Include In Material Report, set the Material Name to "Pavement", set the first subgrade name to "Pavement", and set the Depth to 1.5. Then pick OK.



To save the subgrade changes, pick the Save button on the Define Layer Targets dialog. Then choose Exit.

Now let's visually verify the subgrade areas. In the Inquiry menu, run Subgrade Areas->Hatch Subgrade Areas. There is a dialog to select which subgrade to hatch. Choose the Pavement. Then there is a dialog for the hatch pattern and color. Change the color to green and click OK. Then run Hatch Subgrade Areas again. This time choose Pad and set the hatch pattern to Hex with blue color. The resulting hatch areas show where the subgrade is applied. Notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. When finished viewing the subgrade areas, run Inquiry->Subgrade Areas->Erase Subgrade Hatches.

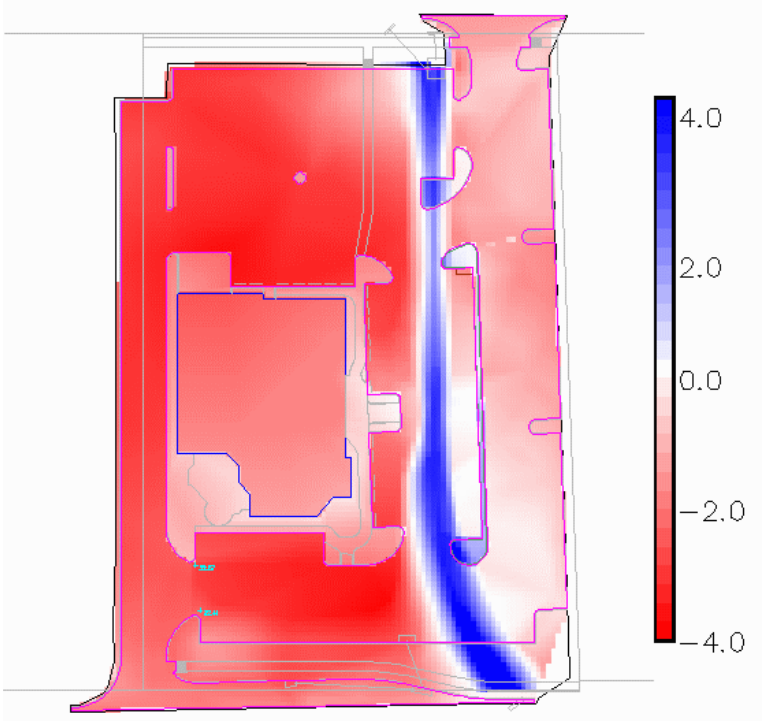


Step 10 (Model Existing and Design Surfaces):

To calculate volumes, Takeoff needs two surfaces: existing ground and design. These surfaces are modeled by triangulation. With the preparation of the previous steps, we're now ready to make the models. To make the existing ground surface, run Tools->Make Existing Ground Surface. The program will process the entities and make the triangulation surface. Then to make the design surface, run Tools->Make Design Surface.

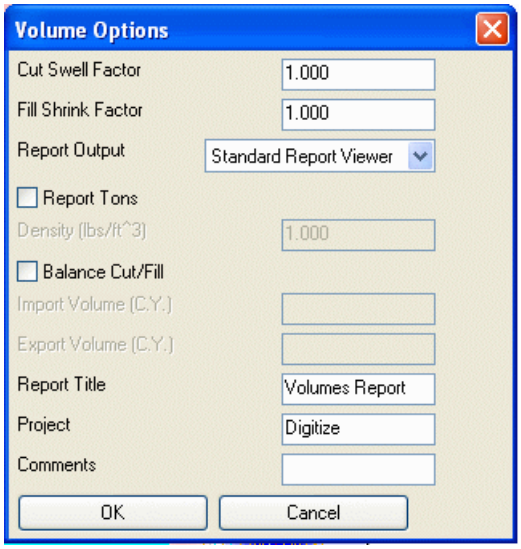
Step 11 (Cut/Fill Color Map):

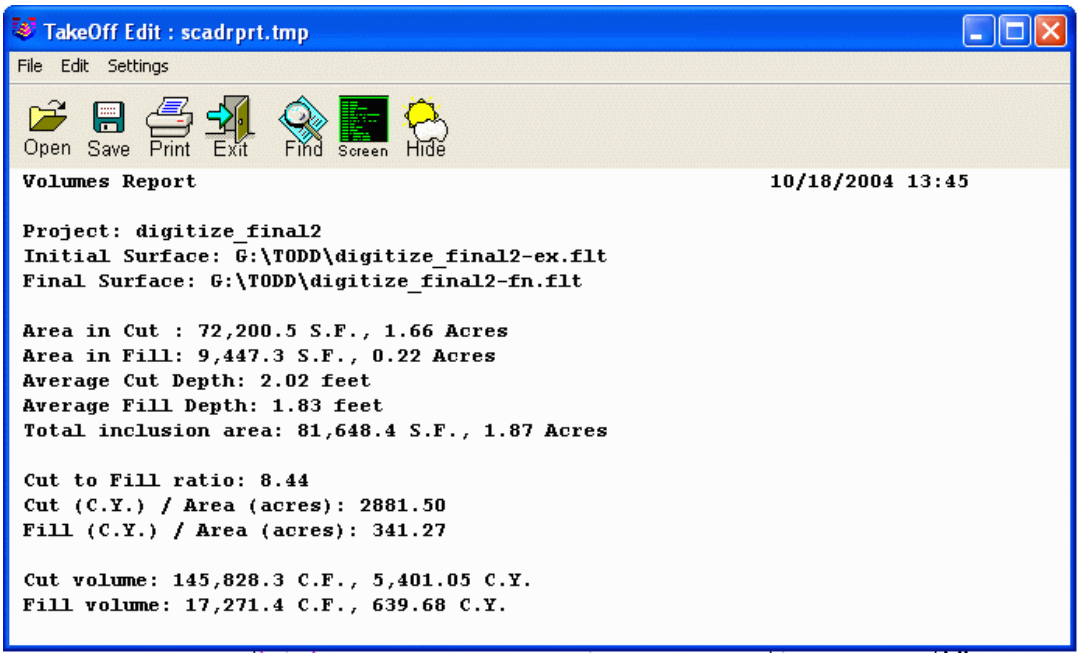
Cut/Fill color maps can be used for a visual output of the site cut/fill areas and also serves as a check that the models are correct. In the Display menu, choose Cut/Fill Color Map. Cut areas are drawn in different shades of red for different depths of cut while fill areas are drawn in blue. To change the resolution of the color blocks, run Display->Display Options and change the Cut/Fill Color Map Subdivisions. This parameter is the number of rows and columns of color blocks to create. You can also draw a legend for the Color Map by going to Draw, Cut/Fill Map Legend. Pick a point on your drawing to locate the legend and press Enter. To turn off the color map, go to the Display menu and pick Cut/Fill Color Map to uncheck it.



Step 12 (Calculate Volumes):

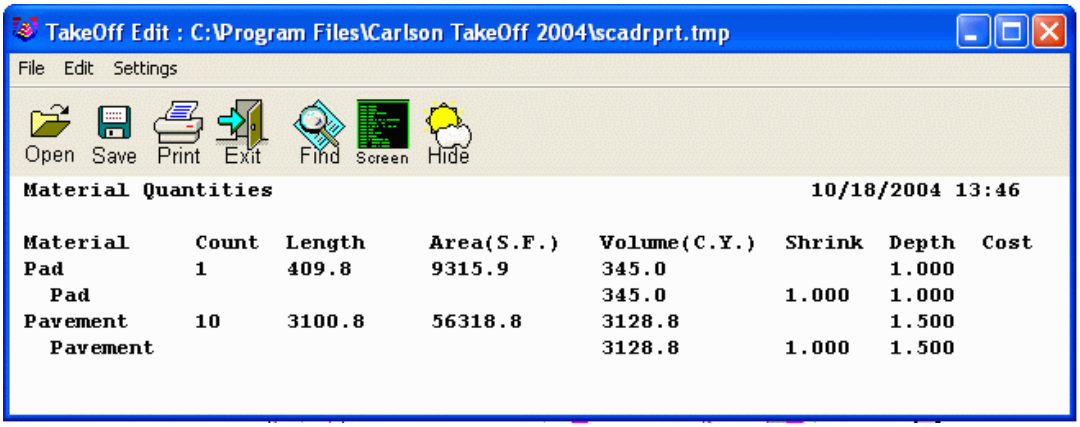
To calculate volumes, run the Tools->Calculate Total Volumes command. There is an options dialog for setting the cut swell factor and fill shrink factor. These values get multipled into the cut/fill volumes. Set these factors as desired and click OK. Then the routine calculates the volumes and display the report which includes the cut/fill volumes and areas. The report can be printed or saved to a file. Pick the Exit button to exit the report viewer.




TakeOff Edit : scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 10/18/2004 13:45
Project: digitize_final2
Initial Surface: G:\TODD\digitize_final2-ex.flt
Final Surface: G:\TODD\digitize_final2-fn.flt
Area in Cut : 72,200.5 S.F., 1.66 Acres
Area in Fill: 9,447.3 S.F., 0.22 Acres
Average Cut Depth: 2.02 feet
Average Fill Depth: 1.83 feet
Total inclusion area: 81,648.4 S.F., 1.87 Acres
Cut to Fill ratio: 8.44
Cut (C.Y.) / Area (acres): 2881.50
Fill (C.Y.) / Area (acres): 341.27
Cut volume: 145,828.3 C.F., 5,401.05 C.Y.
Fill volume: 17,271.4 C.F., 639.68 C.Y.

Step 13 (Material Quantities):

To report the material quantities, run the Tools->Material Quantities->Standard Report routine. The report includes the count, length, area and volume for each type of material that was assigned for reporting in the Define Layer Target/Material/Subgrade command. The Material Quantities->Custom Report routine can be used to reporting these values with control of the report format and the option to export to Excel.


TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Material Quantities 10/18/2004 13:46
Material Count Length Area(S.F.) Volume(C.Y.) Shrink Depth Cost
Pad 1 409.8 9315.9 345.0 1.000
Pad 345.0 1.000 1.000
Pavement 10 3100.8 56318.8 3128.8 1.500
Pavement 3128.8 1.000 1.500

AutoCAD Overview 2

Issuing Commands

Carlson Takeoff is build on the AutoCAD engine. This section will review some basics of AutoCAD that will be useful in running Carlson Takeoff.

Virtually all commands in AutoCAD have three or more ways they can be initiated. The two most common are the menu (if you can find the command you want) and the toolbar (those cute little buttons), but the command line (DOS lives!) can many times be a very easy method of working through commands. Using the menu and toolbar should be second nature to Windows users, so this section will mainly focus on the command line usage in AutoCAD.

Command Line Prompt-Command:

AutoCAD has a command line prompt where commands are "issued" and the status of a command is reported. When you select a command from the menu or toolbar, that command name is sent to the command line and is executed. Before most commands can begin, all other commands must be terminated. The exceptions are referred to as transparent commands. The easiest way to see if no commands are running is to look at the command line. If it displays Command: no commands are currently in progress.

Enter

When you are required to press the Enter key in AutoCAD, you can use the Enter key on the keyboard or you can press the spacebar or click the right mouse button.

Pressing the Enter key will perform different operations depending on your location within AutoCAD. If you are in the process of running a command, the Enter key will end the command (if there are no available options) or it will select one of the options available (see below). If you are at the command line Command: prompt, pressing the Enter key will repeat the last command.

Right Mouse Button

As mentioned above, the right mouse button can be used as enter. However, with AutoCAD 14 and 2000, you can set the right mouse button to perform different functions. When you click the right mouse button, you will get a drop down list of the options and can select the desired one. Move the cursor the desired option and click on it with the left mouse button.

Getting Out of a Command-Esc

For commands that provide no options, the Enter key (or spacebar or right mouse button) will end the command you are using. Also, when a command is issued in AutoCAD, this action will also automatically end the previous command. However, there are exceptions to these two rules.

To avoid any problems with using the above methods to end a command, you can press the Escape (**Esc**) key. For example, if the command line displays something other than Command: or if a command you want to run does not start because you are using another command,you will want to end the command and return to the command prompt. Using the Escape (Esc) key will accomplish this every time.

Note: Some commands have several optional levels which will require the escape key to be pressed more than once.

If you accidentally select a location on the screen and start a selection window, the command prompt will prompt for another corner. Either select another point on the screen to finish the selection (not advised) or press the **Esc** key to return to the Command: prompt.

Commands Option

When any command is issued, the command line acts as a status bar that will show the available options and "ask" for input from you.

When there are options for the command you are currently running, these options will be shown on the command line with capital letters in the option name. To use one of the options, type the capital letter(s) at the prompt. For example, if you issue the Zoom command, the command prompt will show All/Center/Dynamic/Extents/...<Realtime>: To select the Dynamic option, type**D** at the command line, then press the Enter key. If you do not input an option and just hit enter, you will be selecting the option that appears in the <> brackets. For the example shown, that is the Realtime Zoom option.

Transparent Commands

Several commands in AutoCAD can be run transparently. This means that they can be performed while another command is running. For example, if you are in a command and are trying to select something in the drawing but it is too small to see, you can use the zoom command transparently. Zoom to the area where the object is, then select the object without ending the initial command. The most commonly used commands are the View commands of Zoom and Pan, and the Properties commands including the Layer dialog box.

To issue a transparent command, type an **'** (apostrophe) before the command name. For example, **'Z** would be transparent zoom. Note that many commands will automatically be assumed to be transparent if they are issued from the toolbar while another command is running.

When you are in a command that is running transparently, this will be indicated with a >> at the far left of the command line preceding any options or other text. When a transparent command iscomplete, you will return to the command you were previously running. If you are in a transparent command and want to end the command to get back to the command prompt, you will need to press the **Esc** key twice.

Note: If you select a View command while running another command, the other command will not end. You will be running the View command transparently. This is one of the exceptions to the rules for ending a command. If you do not want to run the View command transparently, you need to complete the other command or end it by pressing the **Esc** key.

Selection of Items

Most commands in AutoCAD require the selection of objects. When you need to select objects, the command line will prompt Select objects:. When you are at this prompt, your next step will be to create a selection set. While creating the selection set, the prompt Select Objects: repeats and you can continue to select objects until you press the Enter key, at which time the command you are using will continue and use the objects selected. If you are selecting object for an Eagle Point Software command, the objects selected will be then used for that command.

Selection Sets

There are several ways to create a selection set from the Select objects: prompt. With all selection methods, the number of objects selected will be displayed in the command line along with any objects that were duplicated. Following are the most commonly used methods for creating a selection set:

Single

A single selection is made when you move the object selection target to an object on the screen and click on it. The selected object will highlight and the select objects prompt will return. The cursor changes to a small square when

the command line displays Select objects:.

Window

A Window will select all objects completely inside of the rectangle drawn. Create a window by selecting a point on the view screen and then moving the cursor right. The window will display as a solid rectangle. You can also create a window by typing **W** at the select object prompt. In this case you can move the cursor to the left to create the window.

Crossing

A crossing will select all objects within the rectangle as well as those touched by the rectangle. If you select a point on the view screen and move the cursor to the left, you are creating a crossing. The crossing will display as a dashed rectangle. You can force a crossing by typing **C** at the select objects prompt, allowing you to move the cursor to the right and create the crossing.

Previous

After you select several objects, they will be temporarily stored as a selection set. Should you want to re-use the same objects that were selected by the last command, you can type **P** at the Select objects: prompt.

Remove

If you select incorrect objects, you can type **R** to remove objects from the selection set. When you are in Remove mode, the prompt will be Remove objects:. Click on the objects that you do not want to include in your selection. To return to select or add mode, type **A** at the command line.

Grips

Objects may also be selected before issuing the command and receiving the **Select objects:** prompt. This selection will turn on AutoCAD "grips." Grips appear as small blue squares in the drawing. All objects with grips will be used when a command is issued. Grips are turned on similar to selecting objects as described in the previous section, but with the command line at the **Command:** prompt, not the **Select Objects:** prompt. You can select single objects by clicking on an object with the standard cursor or you can select multiple objects by clicking in the view where there are no objects, then creating the window (right) or crossing (left). To remove grips, press the **Esc** key twice.

Using Grips

Grips can also be used to edit or change the location of objects in the drawing. Move your cursor to a grip and click. You should notice the cursor "snap" to the grip. The grip will turn from blue to red. The object grip is now attached to the cursor so you can move the object to the desired location and release it by clicking again.

General Commands

Enter

When you are required to press the Enter key in AutoCAD, you can use the Enter key, the spacebar on the keyboard or click on the right mouse button.

Repeating Commands

When you press the Enter key at the command line Command: prompt, you will repeat the last command.

Cancel

The Escape key (**ESC**)key can be used to cancel any command. Some commands may require pressing the Escape key more than once.

Command Options

The command line changes as a command is running. When there are options available for the command you are running, they will display at the command line. To select one of the options, type the capital letter(s) in the name of the option and press the Enter key.

Properties and Layers

Properties define how an object in AutoCAD is stored. One of the most common properties is the Layer. Layers can be turned off or frozen so the objects on that layer are hidden from the view of the drawing. Layers that are turned off can still be selected while frozen layers are essentially removed from the working set of the drawing. After layers are turned off or on, a redraw (which is done automatically) will update the view. A redraw of a file is a rather quick process. After a frozen layer is thawed, a regeneration may be required to update the view. Regenerations on large files may take a considerable amount of time depending on your hardware.

Layer Dialog Box

The Layer dialog box provides control of the drawing layers. You can turn layers on, off, freeze or thaw them, change the layer color and linetype, set the current layer, add new layers, delete layers, etc. To perform any of these functions, click on the **Layers**button  .

In the Layer dialog box, you can highlight several individual layers to perform actions on at once by holding down the Control key and clicking on the desired layers. You can also highlight a continuous range of layers by highlighting one layer, holding down the Shift key, and clicking on another layer. All layers between the two will be selected. To select all of the layers, hold down the CTRL key and press A on the keyboard.

Layer Drop List

To turn layers on/off or freeze/thaw, you can also use the drop list on the toolbars. When you click on the symbols in the list, the layer's status will be changed appropriately.

Setting Current Layer

The current layer will be the one shown in the Layer drop list box. You can change the current layer by selecting the desired current layer from the drop list. You can also use the Set Current Layer button and select an object on the layer. The layer the object is on will become the current layer. Finally, you can highlight a layer in the Layer dialog box and click on the **Current** button to make the highlighted layer current.

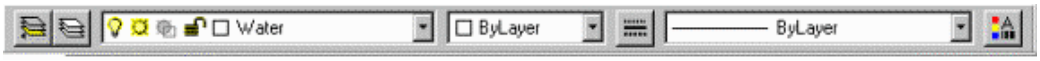
You cannot freeze the current layer, but you can turn that layer off (not recommended).

Changing Properties

To change the properties of an object in the drawing, use the Properties button on the toolbar or the Change command from the command line. This command will allow you to change the layer an object is on, the color or linetype of

the object. The color and linetype can be set to bylayer or to a specific setting. Bylayer means that when the property for a layer is changed, so does the property for the object. For example, the specific setting of an object may be the color blue. No matter what color you set for the layer the object is on, that object will be blue.

Properties



If this toolbar (or any toolbar) is not displayed, you can open it using the Toolbars dialog box. Type **toolbars** at the command line.

Layer

The Layer dialog box is used to modify layers properties (color or linetype) or status (on/off, freeze/thaw).

Key-in: **LA** or **ddlmodes**

Toolbar: 

Current Layer

The Current Layer is the layer that you are on and will be working with. The Current Layer is the one shown in the Layer drop list. For example, in the above toolbar illustration, the Current Layer is **Water**.

Toolbar: 

Change

The Change command allows you to modify the properties of an object, such as layer and color.

Key-in: **CH**

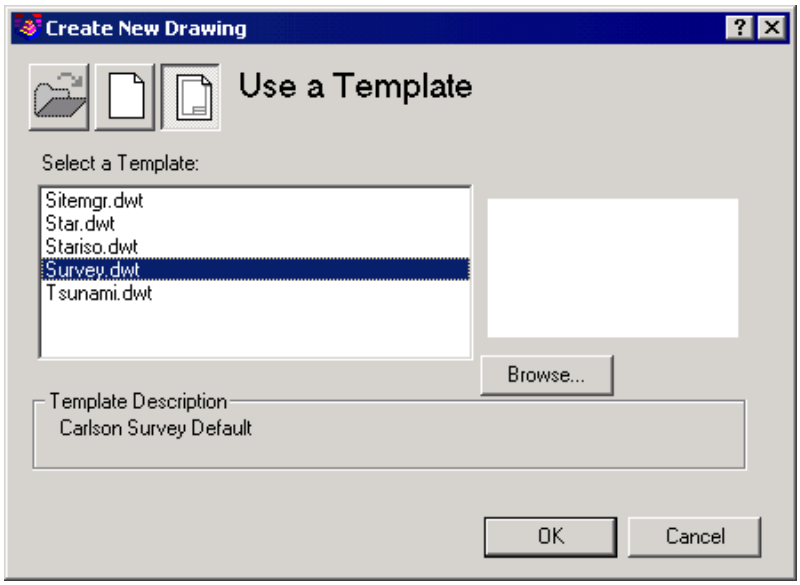
Toolbar: 

File Menu 3

New

Function

This command allows you to create a new drawing file.



This command defines the settings for a new drawing. There are two methods that you can use to create a new drawing. (The first option, Open a Drawing, is not available from the NEW command. To open an existing drawing, use the OPEN command.) Choose one of the icons at the top of the dialog box.

1 Under Start from Scratch, you can start a new drawing file.

This command starts a new drawing using default settings defined in either the surv.dwt or surviso.dwt template, depending on the measurement system you've chosen. You cannot modify the surv.dwt or surviso.dwt templates. To start a new drawing based on a customized template, see Use a Template.

- English: This option starts a new drawing based on the Imperial measurement system. The drawing is based on the surv.dwt template, and the default drawing boundary (the drawing limits) is 12 × 9 inches.
- Metric: This option starts a new drawing based on the metric measurement system. The drawing is based on the surviso.dwt template, and the default drawing boundary (the drawing limits) is 429 × 297 millimeters.

2 Under Use a Template, you can start a new drawing based on a customized template.

This command creates a new drawing using the settings defined in a template drawing you select. Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the .dwt file extension. They are normally kept in the template directory. Several template drawings are included with Carlson TakeOff. You can make additional template drawings by changing the extensions of drawing file names to .dwt.

- Select a Template: This option lists all template files that currently exist in the drawing template file location, which is specified in the Options dialog box. Choose a file to use as a starting point for your new drawing. A preview image of the selected file is displayed to the right.
- Browse: This option displays the Select a Template File dialog box (a standard file selection dialog box) where you can access template files in other directions.

Prerequisite: None

Keyboard Command: NEW

Open

Function

This command allows you to open an existing drawing file. Carlson TakeOff displays the Select File dialog box (a standard file selection dialog box). Select a file and click Open.

Prerequisite: None

Keyboard Command: OPEN

Close

Function

This command allows you to close the current drawing. Carlson TakeOff closes the current drawing if there have been no changes since the drawing was last saved. If you have modified the drawing, the program prompts you to save or discard the changes. You can close a file that has been opened in Read-only mode if you have made no changes or if you are willing to discard changes. To save changes to a read-only file, you must use the SAVEAS command.

Prerequisite: None

Keyboard Command: CLOSE

Save

Function

If the drawing is named, Carlson TakeOff saves the drawing without requesting a file name. If the drawing is unnamed, the program displays the Save Drawing As dialog box (see SAVEAS) and saves the drawing with the file name you specify. If the drawing is read-only, use the SAVEAS command to save the changed file under a different name. This command allows you to save the drawing under the current file name or a specified name

Prerequisite: None

Keyboard Command: SAVE or QSAVE

Save As

Function

This command allows you to save an unnamed drawing with a file name or renames the current drawing.

Carlson TakeOff displays the Save Drawing As standard file selection dialog box. Enter a file name and type. You can select any of the following file types:

- Carlson Software 2002/AutoCAD 2000 (∗.dwg)
- AutoCAD R14/LT 98/LT 97 Drawing (∗.dwg)

- AutoCAD R13/LT 95 Drawing (*.dwg)
- Drawing Template File (*.dwt)
- Carlson Software 2002 DXF (*.dxf)
- AutoCAD R14/LT 98/LT 97 DXF (*.dxf)
- AutoCAD R13/LT 95 DXF (*.dxf)
- AutoCAD R12/LT2 DXF (*.dxf)

Carlson TakeOff saves the file under the specified file name. If the drawing is already named, the program saves the drawing to the new file name. If you save the file as a drawing template, the program displays the Template Description dialog box, where you can provide a description for the template and set the units of measurement.

Saving a drawing in Release 14/LT 98/LT 97 format is subject to the following limitations:

- Hyperlinks are converted to Release 14 attached URLs.
- Database links and freestanding labels are converted to Release 14 links and displayable attributes.
- Database attached labels are converted to MText and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.
- Lineweight information is not available. Lineweights are restored if you open the drawing in AutoCAD 2000 or later.

Saving a drawing in Release 13/LT 95 format is subject to the following limitations:

- Lightweight polyline and hatch patterns are converted to R13 polylines and hatch patterns.
- Raster objects are displayed as bounding boxes. Raster objects are restored if the drawing is opened in AutoCAD 2000 or later.
- Draw order information is not applied for display or print.
- Xrefs that have been clipped with a boundary box are displayed in full as attached xrefs because Release 13 does not support xref clipping. Clipping is restored if the drawing is opened in AutoCAD 2000 or later.

Saving a drawing in Release 12/LT 2 DXF format is subject to the following limitations:

- Lightweight polylines and hatch patterns are converted to R12 polylines and hatch patterns.
- All solids, bodies, regions, ellipses, leaders, multilines, rays, tolerances, and xlines are converted to lines, arcs, and circles as appropriate.
- Groups, complex linetypes, OLE objects, and preview images are not displayed.
- Many objects are lost if you save a drawing as Release 12 and open it later in AutoCAD 2000 or later.

Prerequisite: None

Keyboard Command: SAVEAS

Plot

Function

This command allows you to plot a drawing to a plotting device or file.

Carlson TakeOff displays the Plot dialog box. Choose OK to begin plotting with the current settings and display the Plot Progress dialog box.

1 The Plot dialog box includes the tabs, Plot Device and Plot Settings, and several options to customize the plot.

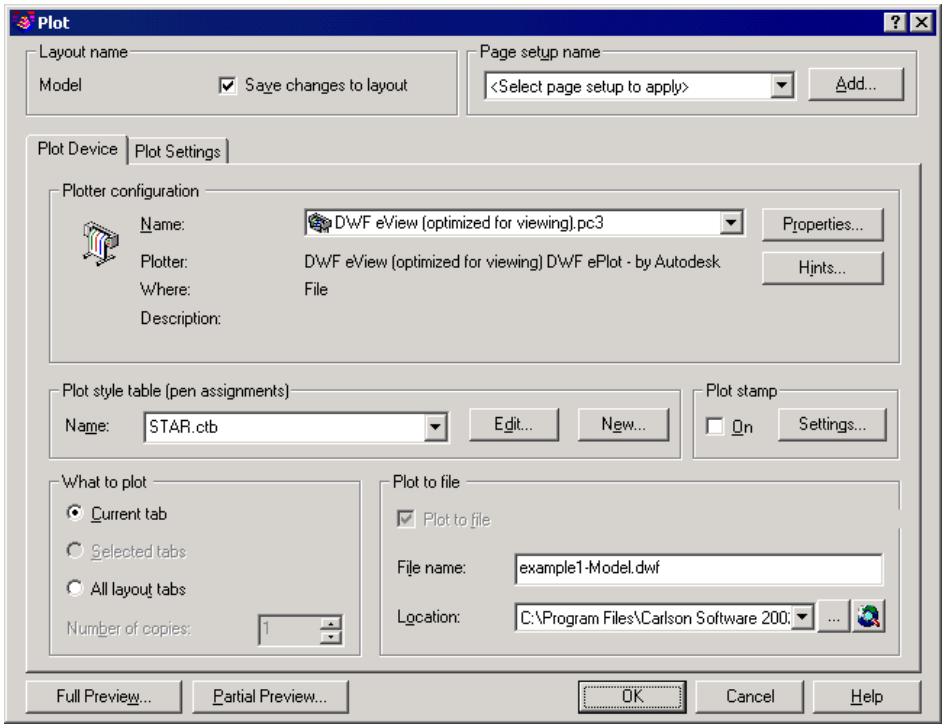
• Layout Name: This option displays the current layout name or displays "Selected layouts" if multiple tabs are selected. If the Model tab is current when you choose Plot, the Layout Name shows "Model."

• Save Changes to Layout: This option saves the changes you make in the Plot dialog box in the layout. This option is unavailable if multiple layouts are selected.

• Page Setup Name: This option displays a list of any named and saved page setups. You can choose to base the current page setup on a named page setup, or you can add a new named page setup by choosing Add.

• Add: This option displays the User Defined Page Setups dialog box. You can create, delete, or rename named page setups.

2 Under the Plot Device Tab you can specify the plotter to use, a plot style table, the layout or layouts to plot, and information about plotting to a file.



• Plotter Configuration: This field displays the currently configured plotting device, the port to which it's connected or its network location, and any additional user-defined comments about the plotter. A list of the available system printers and PC3 file names is displayed in the Name list. An icon is displayed in front of the plotting device name to identify it as a PC3 file name or a system printer.

• Properties: The option displays the Plotter Configuration Editor (PC3 Editor), where you can modify or view the current plotter configuration, ports, device, and media settings.

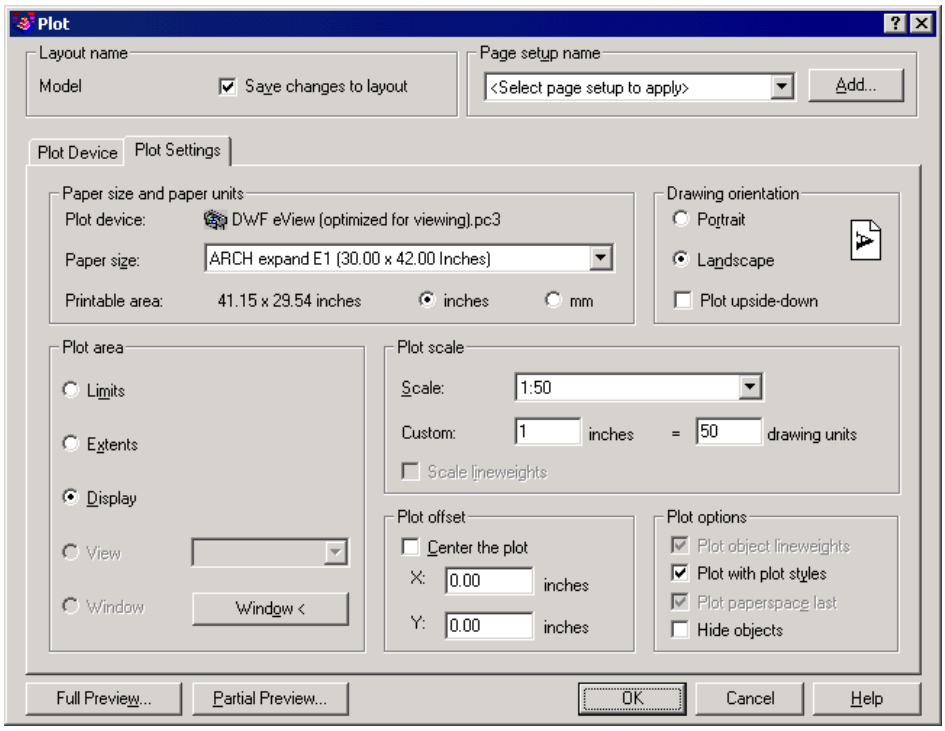
• Hints: This option displays information about the specific plotting device.

• Plot Style Table (Pen Assignments): This option sets the plot style table, edits the plot style table, or creates a new plot style table.

• Name: This option displays the plot style table assigned to the current Model tab or layout tab and a list of the currently available plot style tables. If more than one layout tab is selected and the selected layout tabs have different plot style tables assigned, the list displays "Varies."

- Edit: This option displays the Plot Style Table Editor, where you can edit the selected plot style table.
- New: This option displays the Add-a-Plot-Style-Table wizard, which you can use to create a new plot style table.
- Plot Stamp: This option places a plot stamp on a specified corner of each drawing and/or logs it to a file.
- On: This options turns on plot stamping.
- Settings: This option displays the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, and plot scale.
- What to Plot: This field defines the tabs to be plotted.
- Current Tab: This option plots the current Model or layout tab. If multiple tabs are selected, the tab that shows its viewing area is plotted.
- Selected Tabs: This option plots multiple preselected Model or layout tabs. To select multiple tabs, hold down CTRL while selecting the tabs. If only one tab is selected, this option is unavailable.
- All Layout Tabs: This option plots all layout tabs, regardless of which tab is selected.
- Number of Copies: This option denotes the number of copies that are plotted. If multiple layouts and copies are selected, any layouts that are set to plot to a file or AutoSpool produce a single plot.
- Plot to File: This option plots output to a file rather than to the plotter.
- File Name: This option specifies the plot file name. The default plot file name is the drawing name and the tab name, separated by a hyphen, with a .plt file extension.
- Location: This option displays the directory location where the plot file is stored. The default location is the directory where the drawing file resides.
- [...]: This option displays a standard Browse for Folder dialog box, where you can choose the directory location to store a plot file.

3 Under the Plot Settings Tab you specify paper size, orientation, plot area and scale, offset, and other options.



• Paper Size and Paper Units: This field displays standard paper sizes available for the selected plotting device. Actual paper sizes are indicated by the width (X axis direction) and height (Y axis direction). If no plotter is selected, the full standard paper size list is displayed and available for selection. A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. The paper size you select is saved with a layout and overrides the PC3 file settings. If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

• Plot Device: This field displays the name of the currently selected plot device.

• Paper Size: This field displays a list of the available paper sizes.

• Printable Area: This field displays the actual area on the paper that is used for the plot based on the current paper size.

• Inches: This option allows you to specify inches for the plotting units.

• MM: This option allows you to specify millimeters for the plotting units.

• Drawing Orientation: This option specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation. You can change the drawing orientation to achieve a 0-, 90-, 180-, or 270-degree plot rotation by selecting Portrait, Landscape, or Plot Upside-Down. The paper icon represents the media orientation of the selected paper. The letter icon represents the orientation of the drawing on the page.

• Portrait: This option orients and plots the drawing so that the short edge of the paper represents the top of the page.

• Landscape: This option orients and plots the drawing so that the long edge of the paper represents the top of the page.

• Plot Upside-Down: This option orients and plots the drawing upside down.

• Plot Area: This option specifies the portion of the drawing to be plotted.

• Layout: This option plots everything within the margins of the specified paper size, with the origin calculated from 0,0 in the layout. Available only when a layout is selected. If you choose to turn off the paper image and layout background on the Display tab of the Options dialog box, the Layouts selection becomes Limits.

• Limits: This option plots the entire drawing area defined by the drawing limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option. Available only when the Model tab is selected.

• Extents: This option plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. TakeOff may regenerate the drawing to recalculate the extents before plotting.

• Display: This option plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.

• View: This option plots a previously saved view. You can select a named view from the list provided. If there are no saved views in the drawing, this option is unavailable.

• Window: This option plots any portion of the drawing you specify. If you select Window, the Window button becomes available. Choose the Window button to use the pointing device to specify the two corners of the area to be plotted or enter coordinate values.

• Plot Scale: This option controls the plot area. The default scale setting is 1:1 when plotting a layout. The default setting is Scaled to Fit when plotting a Model tab. When you select a standard scale, the scale is displayed in Custom.

• Scale: This option defines the exact scale for the plot. The four most recently used standard scales are displayed at the top of the list.

• Custom: This option creates a custom scale. You can create a custom scale by entering the number of inches or millimeters equal to the number of drawing units.

• Scale Lineweights: This option scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of printed objects and are plotted with the linewidth size regardless of the plot scale.

• Plot Offset: This field specifies an offset of the plotting area from the lower-left corner of the paper. In a layout, the lower-left corner of a specified plot area is positioned at the lower-left margin of the paper. You can offset the origin by entering a positive or negative value. The plotter unit values are in inches or millimeters on the paper.

• Center the Plot: This option automatically calculates the X and Y offset values to center the plot on the paper.

• X: This field specifies the plot origin in the X direction.

• Y: This field specifies the plot origin in the Y direction.

• Plot Options: This field specifies options for lineweights, plot styles, and the current plot style table. You can select whether lineweights are plotted. By selecting Plot with Plot Styles, you plot using the object plot styles that are assigned to the geometry, as defined by the plot style table.

• Plot object lineweights: This option plots lineweights.

• Plot with Plot Styles: This option plots using the plot styles applied to objects and defined in the plot style table. All style definitions with different property characteristics are stored in the plot style tables and can be easily attached to the geometry. This setting can replace pen mapping in earlier versions of AutoCAD.

• Plot Paperspace Last: This option plots model space geometry first. Paper space geometry is usually plotted before model space geometry.

• Hide Objects: This option plots layouts with hidden lines removed for objects in the layout environment (paper space). Hidden line removal for model space objects in viewports is controlled by the Viewports Hide property in the Object Property Manager. This is displayed in the plot preview, but not in the layout.

• Full Preview: This option displays the drawing as it will appear when plotted on paper. To exit the print preview, right-click and choose Exit.

• Partial Preview: This option quickly shows an accurate representation of the effective plot area relative to the paper size and printable area. Partial preview also gives advance notice of any warnings that you might encounter when plotting. The final location of the plot depends on the plotter. Changes that modify the effective plot area include those made to the plot origin, which you define under Plot Offset on the Plot Settings tab. If you offset the origin so much that the effective area extends outside the preview area, the program displays a warning.

Prerequisite: None

Keyboard Command: PLOT

Recover

Function

This command opens a drawing file and scans it for errors. Use this command if Carlson TakeOff crashes while using the regular **Open** command.
Prerequisite: none

Keyboard Command: recover

Audit

Function

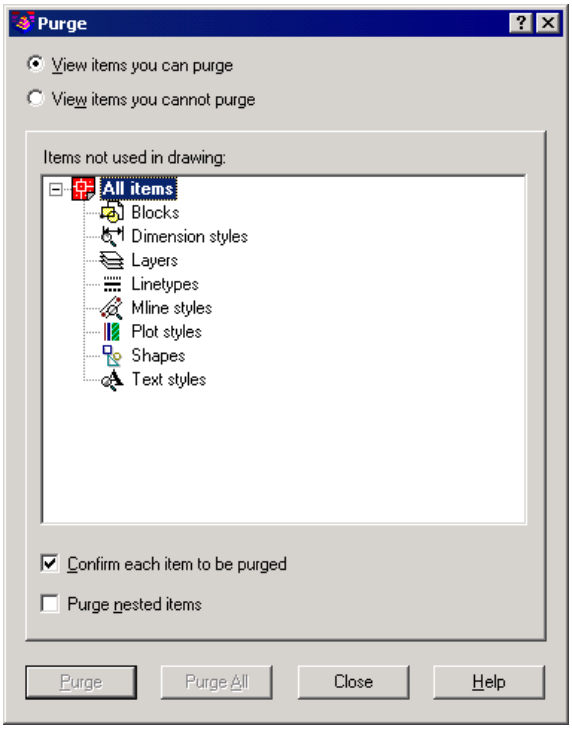
This command scans your current drawing and looks for any corruption and has the option to fix any errors.
Prerequisite: none

Keyboard Command: audit

Purge

Function

Displays a tree view summary of all named objects that can and can't be purged in the current drawing. The View Items You Can Purge and View Items You Cannot Purge options toggle the dialog box display, showing different options and tree view summaries.



Items Not Used in Drawing: Displays a tree view of all named object categories (blocks, layers, and so on) in the current drawing. A plus sign appears next to the object category names that you can purge. Clicking the plus sign or double-clicking an object category expands the tree view, displaying all unused named objects that exist for the category. To purge all unused named objects, select All Items in the tree view, and choose Purge All. To purge a specific named object category, select the category in the tree view, and choose Purge.

Confirm Each Item to Be Purged: Displays the Verify Purge dialog box when you purge an item.

Purge Nested Items: Removes all unused named objects from the drawing even if they are contained within or referenced by other unused named objects. The Verify Purge dialog box is displayed, and you can cancel or confirm

the items to be purged.

Prerequisite: None

Keyboard Command: purge

Store Project Archive

Function

This command creates an archive of the current project. The archive contains the drawing file (.dwg) and all the associated data file such as the surfaces and layer target definitions. This archive can be used as a backup for the project or as a way to transfer the project to another computer. The format of the archive file is a standard .zip file which can be used by WinZip.

When this command is run, the program will ask for a file name of the archive to create. Enter a name and pick the Save button. The number of files stored to the archive is reported at the command line.

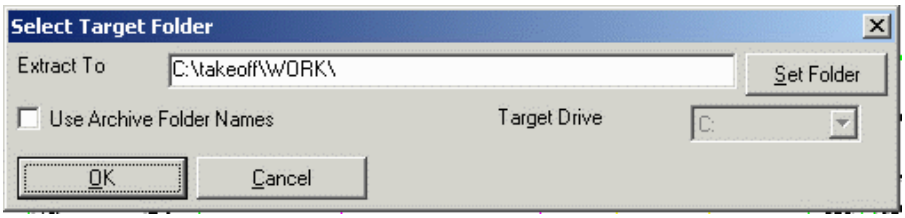
Prerequisite: an active Takeoff project

Keyboard Command: zip_project

Extract Project Archive

Function

This command reads the project files from an archive created by the Store Project Archive command. The archive contains the drawing file (.dwg) and all the associated data file such as the surfaces and layer target definitions. Since the archive contains the drawing file, you should not have the same project drawing open in Takeoff while extracting the archive. The format of the archive file is a standard .zip file which can be used by WinZip.



The command starts by prompting for the archive file to open. Then there is a dialog with extraction options. You can either extract the files to the specified folder or use the folder names stored in the archive. When using the archive folders, there is a setting to choose the target drive because the archive has the folder names but not the drive names.

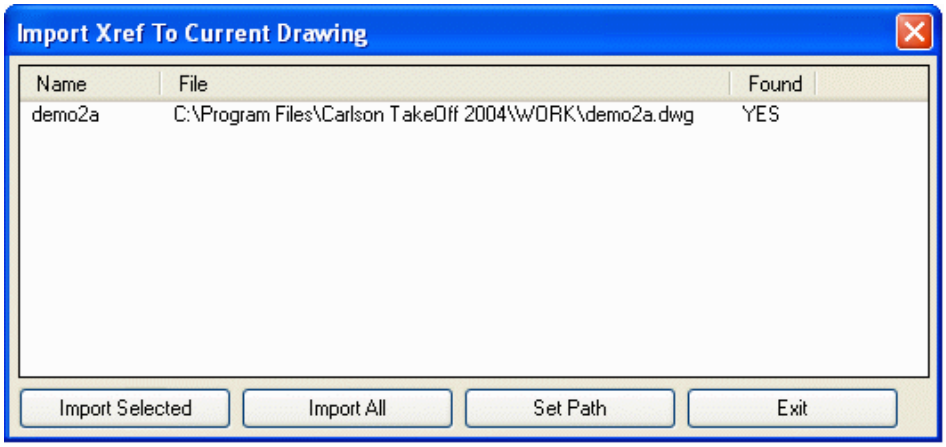
Prerequisite: a Takeoff archive file

Keyboard Command: unzip_project

Import Xref to Current Drawing

Function

This command allows you to import external reference files (Xrefs) into the current drawing. Before Xrefs are imported, the drawing data from the Xrefs can be viewed but not modified. This import routine has a simplier method for importing than the Xref Manager command. A list is shown of the Xrefs that are attached to the current drawing. If the Xref file is not found, you can pick the Set Path button to locate the drawing file. To import an Xref, highlight the file name and Pick Import.



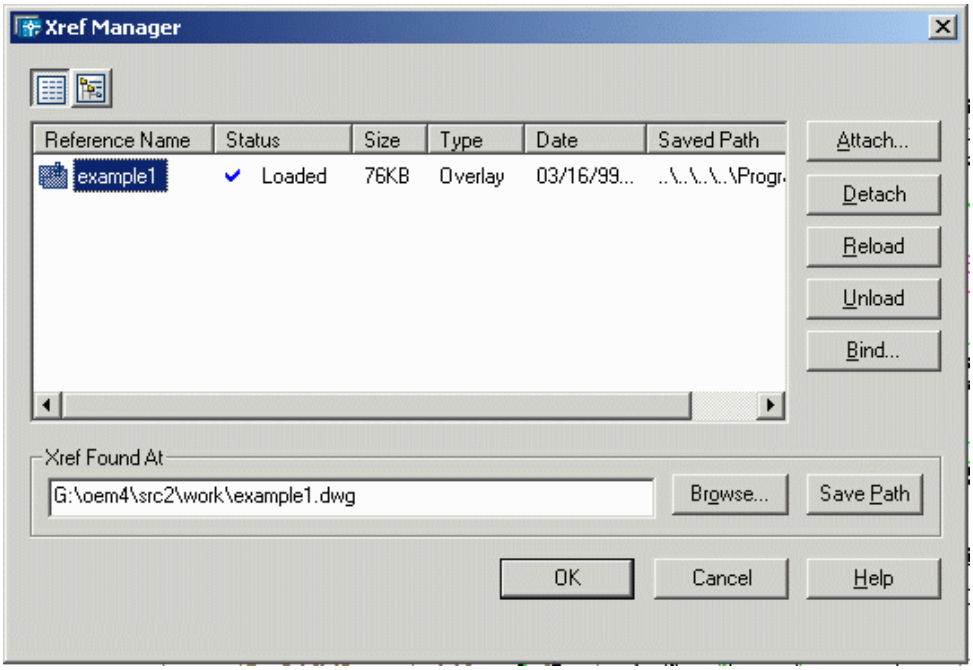
Prerequisite: files to import

Keyboard Command: import_xref

Xref Manager

Function

Attaches, overlays, lists, binds, detaches, reloads, unloads, renames, and modifies paths to external references (Xrefs) in the current (or host) drawing. Displays the Xrefs in the drawing in a tree view or a list view. You can use the F3 and F4 keys to switch between list view and tree view.



List View: Displays a flat listing of the attached Xrefs and their associated data. You can sort the list of references by name, status, type, file date, file size, or the saved path and file name.

Reference Name: Lists the names of the Xrefs as stored in the definition table for the drawing. Status: Shows whether the Xref is loaded, unloaded, unreferenced, not found, unresolved, orphaned, or marked for unloading or reloading.

- Loaded: Currently attached to the drawing.
- Unloaded: Marked to be unloaded from the drawing once the Xref Manager is closed.
- Unreferenced: Attached to the drawing but erased.
- Not Found: No longer exists in the valid search paths.
- Unresolved: Cannot be read by AutoCAD.
- Orphaned: Attached to another Xref that is unreferenced, unresolved, or not found.

Size: Shows the file size of the corresponding reference drawing. The size is not displayed if the Xref is unloaded, not found, or unresolved.
Type: Indicates whether the Xref is an attachment or an overlay.
Date: Displays the last date the associated drawing was modified. This date is not displayed if the Xref is unloaded, not found, or unresolved.
Saved Path: Shows the saved path of the associated Xref (this is not necessarily where the Xref is found).
Tree View: Displays a hierarchical representation of the Xrefs, displaying the relationships between Xref definitions. Tree view shows the level of nesting relationship of the attached Xrefs, whether they are attached or overlaid, and whether they are loaded, unloaded, marked for reload or unload, or not found, unresolved, or unreferenced.

Attach: Displays the External Reference dialog box if an external reference is selected or displays the Select Reference File dialog box if no external reference is selected.
Detach: Detaches one or more Xrefs from your drawing, erasing all instances of a specified Xref and marking the Xref definition for deletion from the symbol table. Only the Xrefs attached or overlaid directly to the current drawing can be detached; nested Xrefs cannot be detached. Carlson TakeOff cannot detach an Xrefreferenced by another Xref or block.
Reload: Marks one or more Xrefs for reloading. This option rereads and displays the most recently saved version of the drawing.

Unload: Unloads one or more Xrefs. Unloaded Xrefs can be easily reloaded. Unlike detaching, unloading does not remove the Xref permanently. It merely suppressesthe display and regeneration of the Xrefdefinition to improve performance.

Bind: Displays the Bind Xrefs dialog box Xref. The Bind option makes the selected Xrefand its dependent symbols (such as blocks, teXreft styles,dimension styles, layers, and linetypes) a part of the current drawing.
Found At: Displays the full path of the currently selected Xref. This is where the Xref is actually found and is not necessarily the same as the savedpath.
Browse: Displays the Select New Path dialog boXref (a standard file selection dialog boXref), in which you can select a different path or filename.

Save Path: Saves the path, as it appears in XrefFound At, to the currently selected Xref.

Prompts

Command: _Xref
Overlay Xref "example1": ..\..\..\..\Program Files\Carlson TakeOff 2004\WORK\example1.dwg
"example1" loaded: G:\oem4\src2\work\example1.dwg

Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:
Command: Specify opposite corner:
Select objects: Enter

Prerequisite: multiple files

Keyboard Command: Xref

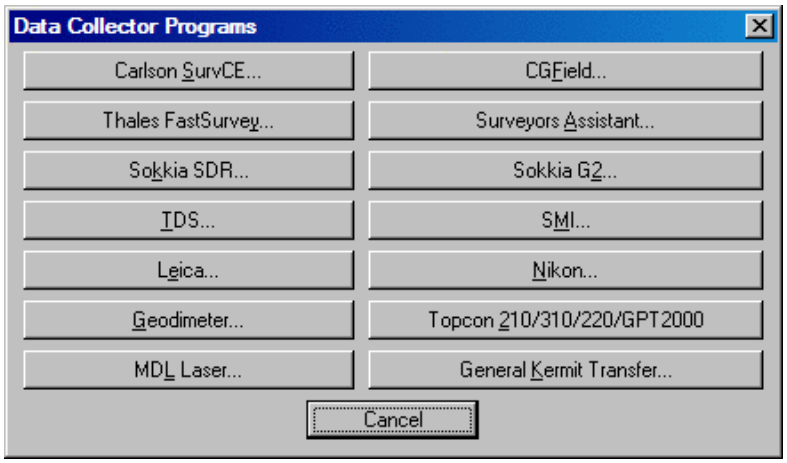
Import-Export

Data Collectors

Function

This command does two main functions for a variety of popular data collectors. First, this command transfers (uploads and downloads) data between the data collector and Carlson. Second, this command converts data formats between the data collector format and the Carlson format. So, if you already have the data file on the computer, you can skip the transfer function and just perform the conversion function.

The transfer function does the conversion at the same time. In most cases, the download from the data collector produces a raw (.RW5) file (field notes) and/or a coordinate (.CRD) file (coordinate points). Several of the download programs have an option to automatically run the *Edit-Process Raw Data File* command after downloading raw data. You can also send, or upload, a coordinate (.CRD) file. The dialog shown here appears when the menu command is selected.



Carlson SurvCE: For Carlson Software data collection programs SurvCE and SurvStar. This button produces the SurvCOM dialog and program.
CG Field: For CG Field programs.
Thales FastSurvey: For Thales and FastSurvey intstruments.
Surveyors Assistant: For data collectors running Surveyors Assistant software (Corvallis MC2, MC5 and Pentax SC5).
Sokkia SDR: For SDR2 through SDR 33 and other collectors that have a SDR format like the Trimble.
Sokkia G2: Specifically for the SDR2.
TDS: For data collectors that use TDS software (Ranger, HP48, HP95, Husky FS-2 & FS-3, Corvallis MC-V and TOPCON FS2, FC95 and FC48).
SMI: For SMI data collectors on the HP48.
Leica: For Leica GIF-10 module and Leica instruments.
Nikon: For Nikon DTM and DR-48 total stations.
Geodimeter: For the Geodimeter Geodat collector.
Topcon 210/310/220/GPT2000: Supports these Topcon models.
MDL Laser: For MDL Laser instruments.
General Kermit Transfer: For general transferring using Kermit.

Carlson SurvCE

Note: In the following text, the term SurvCE will apply to SurvCE, SurvStar, and Sokkia G2

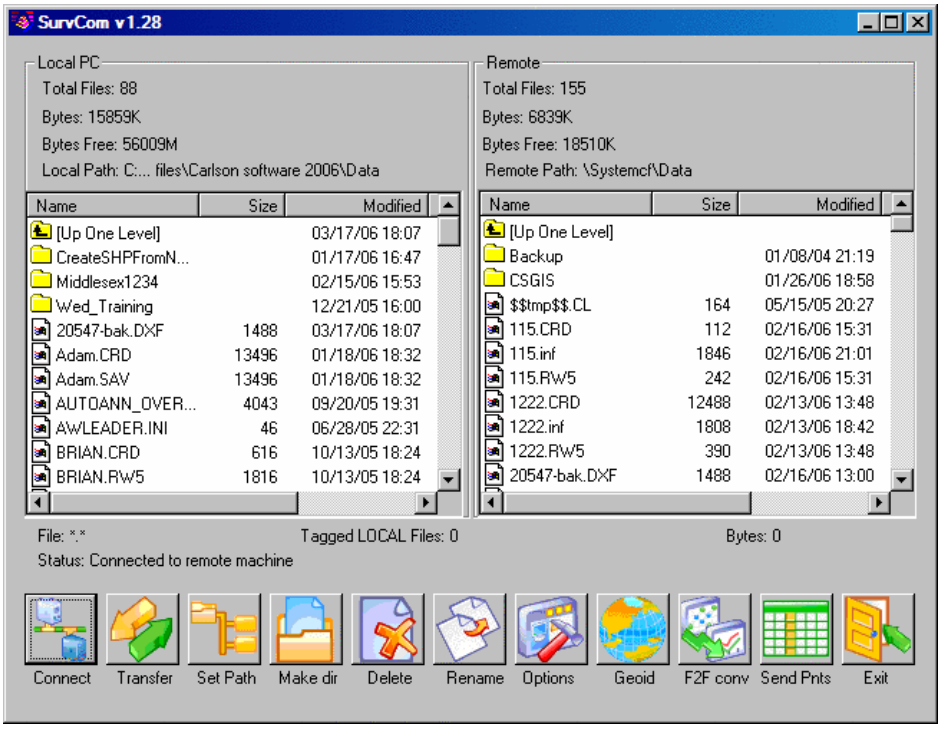
Connect the serial cable. Select Data Transfer from the on the handheld. Choose Carlson/Carlson Survey Download. This leads to a File Transfer screen on SurvCE, which says "Awaiting Connection". All the action is on the PC side. There is no time delay in this handshake. It will wait for the PC program to catch up. When you connect the cable from SurvCE to the PC, Microsoft ActiveSync may interfere and say "Connect to PC?" If you get this question,

say No or otherwise terminate the Microsoft ActiveSync linkage. Start the Carlson portion of this link by choosing Survey, Data Collectors, then the SurvCE option. If connection is automatically established, SurvCE will display, "Connected to PC".

If only the left side of the screen displays data, then you do not yet have a connection. Press the Connect button located at the bottom left of the file transfer dialog. The transfer program will respond with Retrieving File List. Once the file list has been retrieved, the left side of the dialog box will show files located in the specified path on the PC and the right side of the dialog displays the files located in the designated path on the remote. You can change directories by scrolling to the top of the file list and choosing Up One Level (just like in Windows).

To transfer one or more files, simply select or highlight the desired files and select the transfer button. More than one file can be transferred from the remote to the PC or from the PC to the remote during the transfer process. Standard Windows selection options apply. For example, selecting one file and then while pressing the shift key on the PC, selecting another file deeper on the list will select all the files in between the first and last selected. You can also select the first file to transfer and press and hold down the shift key and use the down arrow to specify the range of files to transfer. Pressing and holding the control key on the keyboard allows for the selection of multiple files in any selection order, by picking the files with the left mouse button.

After the files have been selected, press the transfer button. When the transfer is complete, the program will return a "Transfer Complete" message, and will then proceed to update the file lists on the PC and the Remote.



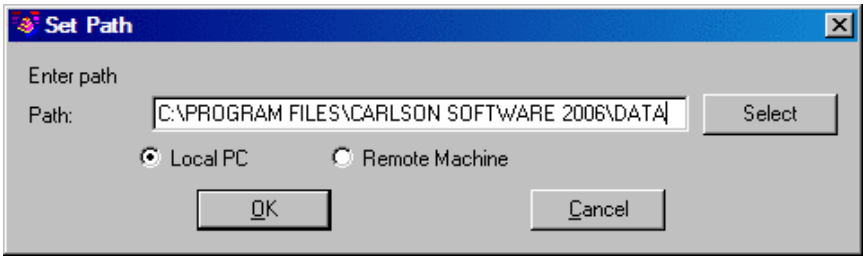
The following information describes the buttons on the bottom row of the SurvCOM dialog box. The button name is on the left in bold:

Connect: After selecting Data Transfer in SurvCE, press this button to start the connection. Once connection is made, the status line on the
file transfer utility dialog box will show Connected to the remote machine.

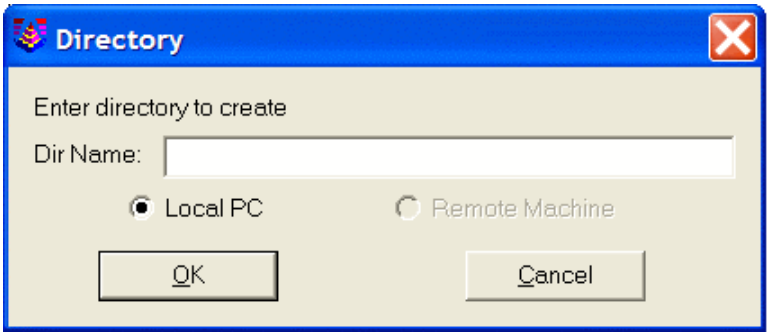
Transfer: Pressing this button transfers selected files from either the Remote to the PC, or the PC to the Remote.

Set Path: This option allows for the specification of the desired source and destination drives and folders for both the PC and the Remote device. For example, if you were downloading, or copying files from the Remote device to

the PC, to specify a source path on the remote device, select the Remote Machine toggle and then type in the desired path in the path field. To specify a destination path on the PC, select the Local PC toggle and type in the desired path the path field. When a change to either path is made, the transfer utility will retrieve a new file list from the specified paths.



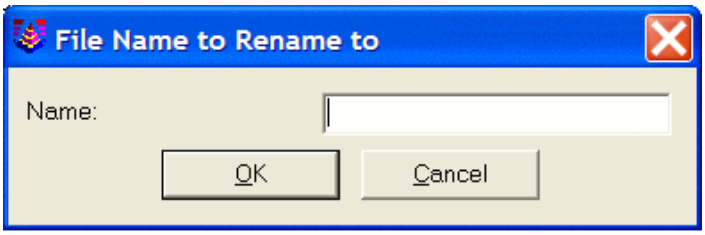
Make dir: This option allows for creation of directories on both the PC and the Remote device. Specify the hardware on which to create the directory and then enter the directory name.



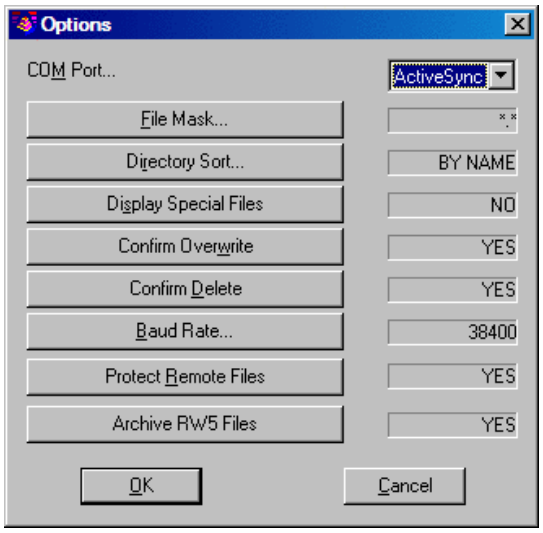
Delete: This option allows you to delete the tagged files. To delete a file, select the file to delete by clicking on the file, press the delete button at the bottom of the dialog. Confirm deletion by selecting the appropriate response on the Delete File dialog.



Rename: To rename a file, click on the file to rename and select the rename button at the button of the dialog. On the dialog that displays type in the new name and press the OK button.

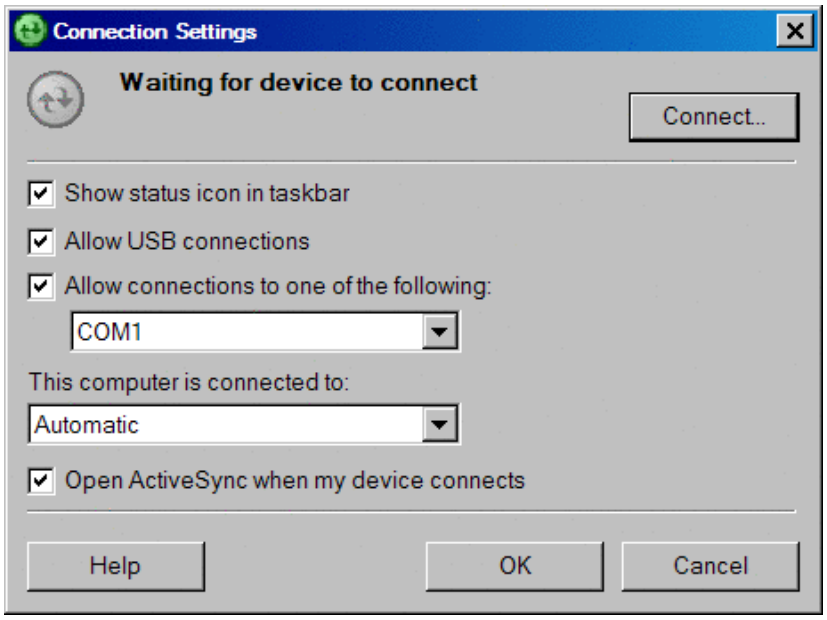


Options: This command allows you to set various options for data transfer. The dialog shown below will appear.



Com Port: You must select which com port on the PC to use.

If you are transferring data via a USB port, set the com port to ActiveSync, see the Options section below for procedures to change com ports. To transfer data using an USB port a connection between the Remote and PC using ActiveSync is required. In ActiveSync verify that the "Connect Settings" have been set to "Allow serial cable or infrared connection to this Com port" and Allow USB connection with this desktop computer. This will allow for connection using an USB port or a COM port connection. Both will use ActiveSync to transfer data between devices.



File Mask: You must select a file filtering syntax. This filter allows for the setting of specific file types to display. For example if you only wanted to see CRD files the filter would be ∗.CRD.
Directory Sort: You must select how to sort the list of files.



Display Special Files: Toggle whether or not you should see special files.
Confirm Overwrite: Check this to confirm before overwriting files.
Baud Rate: You must choose the baud rate for transferring data.
Protect Remote Files: Check this to protect files on the mobile device.
Archive RW5 Files: With this option set to YES, when downloading rw5 files, a second copy of the file will be made with a .SC5 extension to serve as an archive of the original rw5 file.

Geoid: This command will carve out a portion of the Geoid 99, EGM96, Canadian CGC2000, Canadian HT2.0, Canadian HT 1.01, Australian GDA94, Great Britain OSG-MO2 and Geoid 2003 grid files, and send it to SurvCE. Since these geoid grids are very large, this carves out a precise portion of it and avoids overloading the memory on the remote device running SurvCE. You will be prompted for the directory on the PC of the source Geoid grid file, the approximate latitude and longitude of the job, and the size of the area desired in miles, kilometers or degrees of latitude and longitude. To define a Geoid area, make sure that this criteria is met:

1. Specify the location of the geoid grid files.

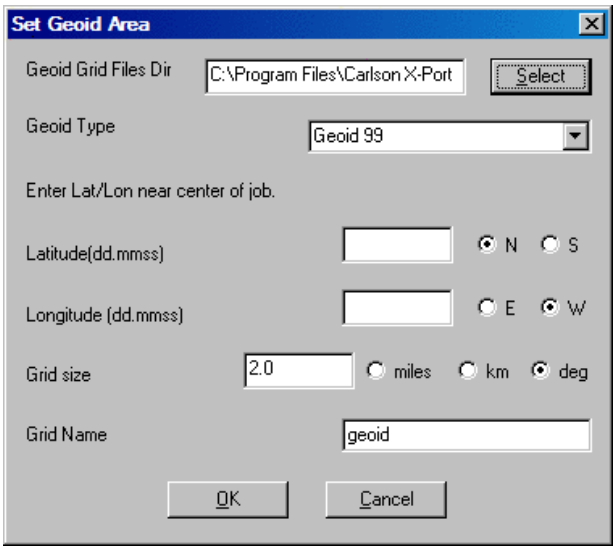
2. Specify the geoid type.

3. Enter the latitude and longitude near the center of the job area.

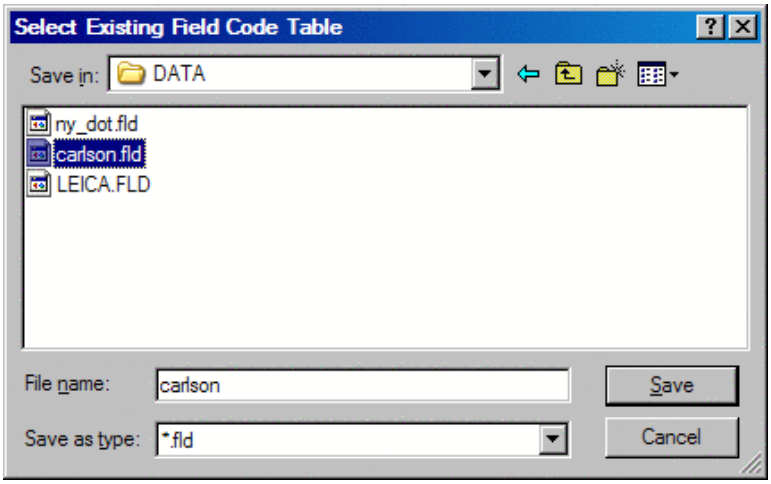
4. Specify the Grid size either in miles, km (kilometers), or deg (degrees).

5. Name the grid file.

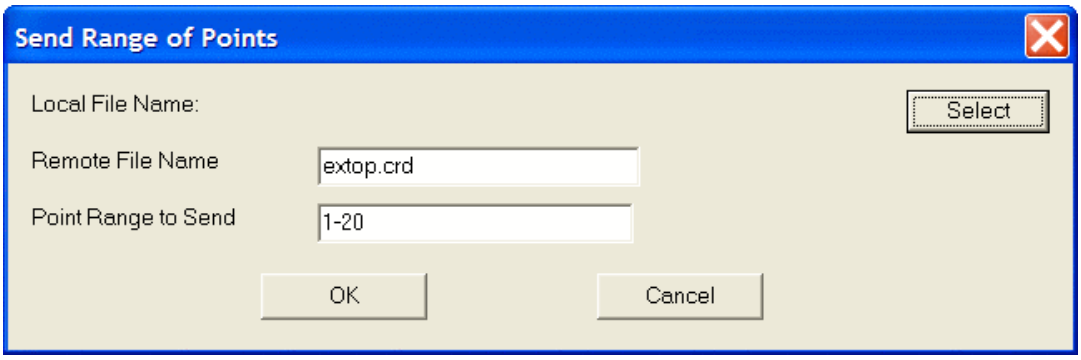
The file will be transferred to the data collector and place in the appropriate place for use.



F2F conv: This converts the more thorough and detailed Carlson field code file (for field-to-finish work,∗.FLD) to the more simplified Feature Code List that runs in SurvCE (∗.FCL). The Feature Code List in SurvCE (not SurvStar or Field) handles Linework (on or off), Line Type (2D or 3D), Layer (= Code) and Full Text (Description). Select the Carlson field code (∗.FLD) to convert, the conversion takes place and the file is transferred and located in the correct location for use in the data collector.



Send Pnts: This option allows for the uploading of a user specified point number range out of the selected crd file to unload. Use the Select button to specify the crd file to upload. The Remote File Name will default to the name of the crd file selected to upload. You can change this name if needed. Specify the Point Range to Send and select the OK button.



Exit: This command will exit the File Transfer Utility

The following information describes the buttons on the Data Collection Programs dialog box that come after the Carlson SurvCE buttom, moving from left to right and then from top to bottom. The command/button name is on the far left margin, in bold:

CG Field

To transfer data to and from data collectors using CGField software, first make sure that the Baud Rate is set to 9600 and the Parity is set to NONE then follow the steps outlined below.

Receiving a Coordinate File from CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 4, "Send Coords"
3) Select the Coordinate file to send.

Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

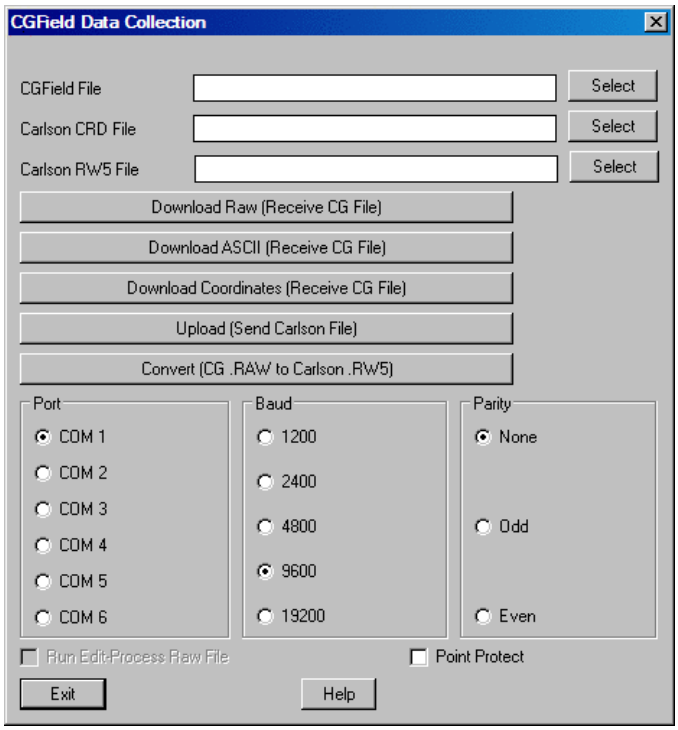
Press the "Download Coordinates" button to ready Carlson to receive the file.
Stop here in Carlson and go back to CGField to complete the transfer process.

CGField:

Select the points to send

1) For All points
2) To select Blocks of points.
3) From .PTS file (the set of points in a Batch Point File).

The coordinates will be transferred. After the transfer is complete, you will be asked for the CRD file name. The C&G CRD file will automatically be converted to a Carlson CRD file. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Receiving a Raw Data File from CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 2, "Send Raw Data". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download Raw" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the raw data file to be sent. The transfer will begin.

The C&G .RAW file will be transferred and saved in the data folder. After the transfer is complete, you will be asked for the RW5 file name. The RAW file will be automatically converted to a Carlson RW5 file.

Receiving an ASCII file from CGField

This will allow you to transfer a C&G report file (RPT) or an ASCII NEZ file to Carlson.

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 6, "Send ASCII". Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

Press the "Download ASCII" button to ready Carlson to receive the file. Stop here in Carlson and go back to CGField.

CGField:

Select the ASCII file to send.

After the transfer is complete, you will see the file in the Carlson editor. You can then select FILE and SAVE (or SAVEAS) to save the ASCII file.

Sending a Coordinate File to CGField

CGField:

1) Go to the UTILS menu and select Option 1, C&G Transfer.
2) Select Option 3, "Receive Coords" to ready the data collector. Stop here in CGField and go to Carlson.

Carlson:

Leave the FILE fields blank.

1) Press the "Upload (Send Carlson File)" button.
2) Select the Coordinate file.
3) Select the points to send.
4) Press the "Start Transfer" button.

CGField:

Carlson will send the file name to CGField and a coordinate file with the same name will be automatically created or opened in CGField.

If the file exists you will be asked how you want to handle duplicate points:

1) Overwrite
2) Don't Overwrite
3) Ask for each Point

The point transfer will begin.

Convert CG .RAW to Carlson .RW5

This utility allows you to convert a C&G raw data file to a Carlson raw data file. Select the C&G .RAW file to convert. Then enter the file name of the destination Carlson RW5 file.

Thales/FastSurvey You will be taken directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

Surveyor's Assistant

Download
From the Surveyor's Assistant data collector, go to the Transfer routine from the main menu. Fill out the transfer screen as follows:

Direction: OUTPUT
Format: LIETZ
Data: Coordinate or All Data
Port: COM1 or COM2 Ckh Hold: NO
Protocol: NONE

You should also check the settings under the PORT menu. Typical port settings are baud=9600, parity=none, data=8, stop=1 and handshake=XON/XOFF. Now in Carlson, run *Data Collection* in the Survey menu and choose Surveyor's Assistant. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to Surveyor's Assistant and press GO. The file transfer should now go. If the All Data option is used, then the Leitz format will contain both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified

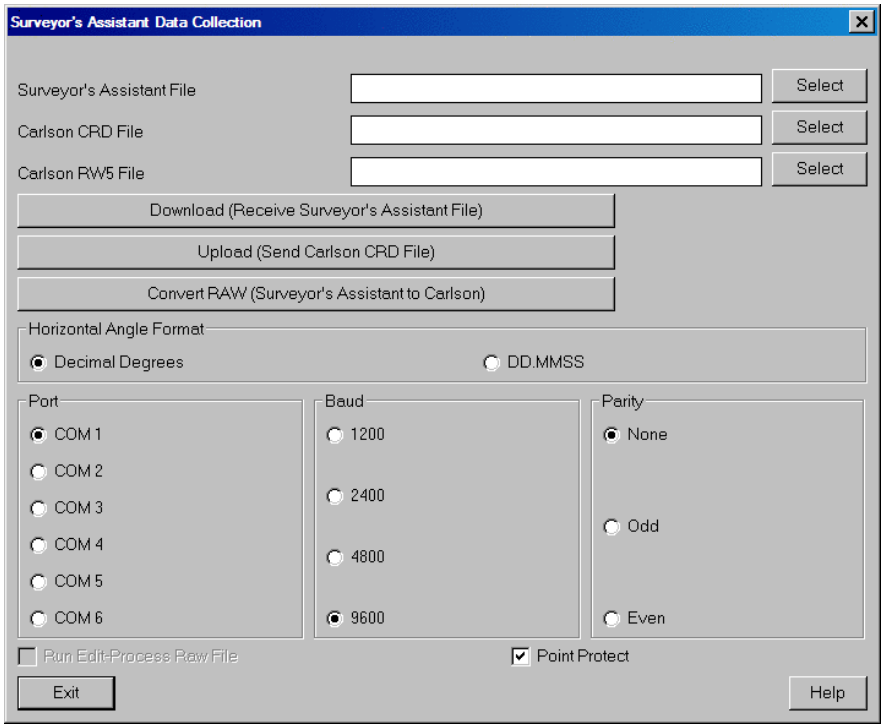
a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

Point data from the Carlson coordinate (.CRD) file can be uploaded into the Surveyor's Assistant. First go to the Transfer routine on the main menu. Fill out the screen as follows:

Direction: INPUT
Format: LEITZ
Port: COM1 or COM2
Protocol: NONE

Go back to Carlson and choose Surveyor's Assistant from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to the Surveyor's Assistant and hit the GO function key. The Surveyor's Assistant is now waiting to receive so return to Carlson and click OK on the range of point dialog. The file transfer should now go.



Sokkia SDR

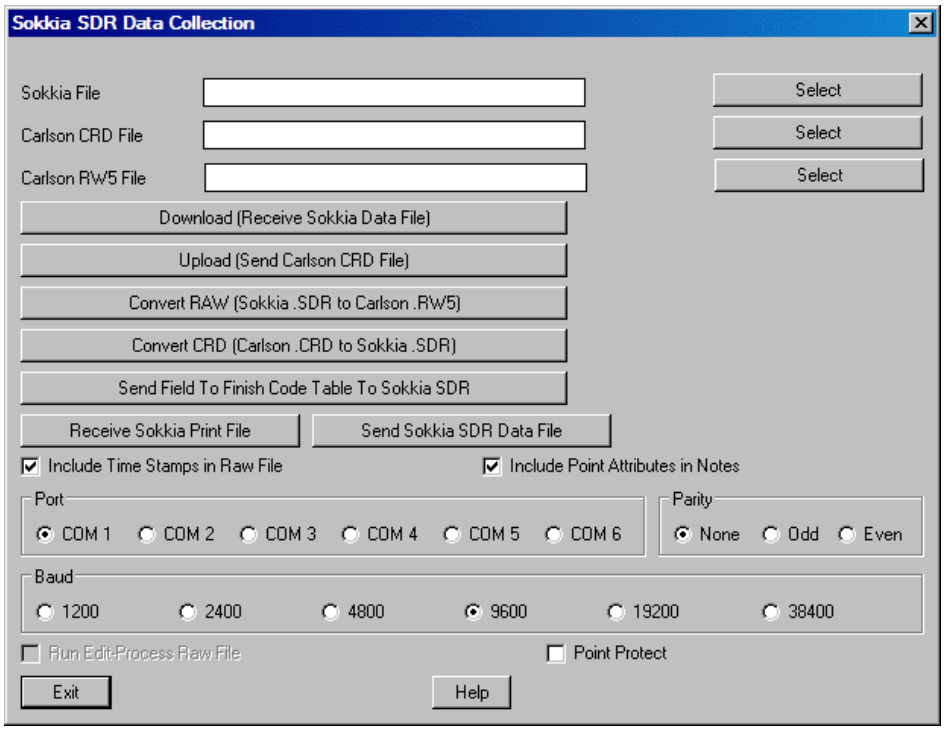
This routine applies to the Sokkia SDR-20, SDR-22, SDR-31 and SDR-33 as well as other collectors that have SDR format transfer such as the Trimble and C & G.

Download

From the SDR data collector, go to the Communications routine from the main menu. Choose Data Format SDR. Next hit the Send function key. Then choose Select Jobs. From the list of jobs, highlight the job to transfer and set it to Yes with the arrow keys. Now in Carlson, run *Data Collection* in the Survey menu and choose Sokkia/SDR.

Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SDR and press OK. The file transfer should now go. The SDR format contains both coordinate and raw data. The coordinate data is converted to a Carlson coordinate (.CRD) file and the raw data is converted to a Carlson raw data (.RW5) file. The original SDR transfer file is stored on the computer as a RAW file. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

The SDR-33 has different modes for storing and transferring data. In coordinate mode, the download will create points in the coordinate (.CRD) file and the raw data (.RW5) file will only contain some basic header lines. In the raw data mode, the download will create all the measurement data in the raw file and no points will be created in the coordinate (.CRD) file. For this raw data mode, you will need to run *Edit-Process Raw Data File* in the Survey menu to calculate the points from the raw data. The third mode in the SDR-33 creates both raw data in the raw data (.RW5) file and points in the coordinate (.CRD) file. The Include Time Stamps in Notes option sets whether all the date-time records for each point are put in the raw data (.RW5) file as description records. The Include Point Attributes in Notes option will store SDR code 13(AT) codes to the point note (.NOT) for the coordinate (.CRD) file.



Upload

Point data from the Carlson coordinate (.CRD) file can be uploaded into the SDR. First go to the Communications routine on the SDR main menu. Choose Data Format SDR. Go back to Carlson and choose Sokkia/SDR from the *Data Collection* command in the Survey menu. Check that the COM port and baud rate are set correctly. In the Carlson dialog, pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Then click the Upload button. Then a Sokkia Options dialog appears for setting the job parameters for the file to be created on the collector. Be sure to choose the Distance Unit that matches your coordinate (.CRD) file (meters, US feet or international feet). Click OK and the next dialog now allows you to specify the range of point numbers to upload. Before clicking the Start Transfer button for range of points, go to the SDR and hit the Receive function key. The SDR is now waiting to receive so return to Carlson and click Start Transfer on the range of point dialog. The file transfer should now go.

Communication Settings
Besides matching the baud rate between Carlson and the collector, make sure that the collector is set to word length of 8 and 1 stop bit under the communication settings.

Print File
The Receive Sokkia Print File downloads a print report from the SDR33 data collector. This file is only used for printing report purposes in Carlson. This file is not used by Carlson to generate coordinate (.CRD) files or raw files. The first step is to choose Data format=Printed in the Communications menu of the SDR33. Next pick the Receive Print File button in Carlson. Then on the SDR33 choose the Send function and select a job to send. At this point the file is transferred. After downloading, the job report is displayed in the Carlson standard report viewer.

Example of Sokkia Printed Format:

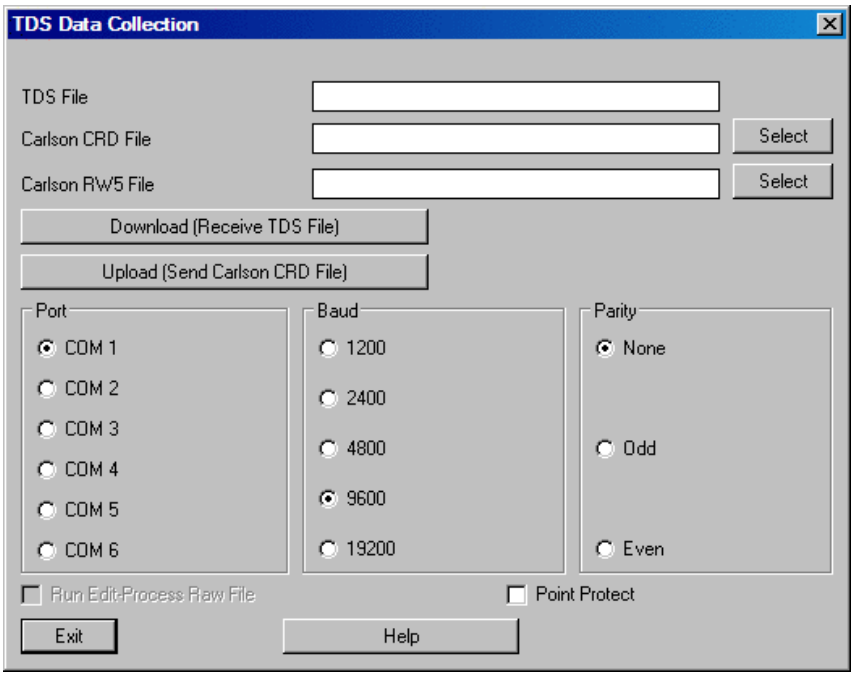
```
SDR33 V04-04.25 (C) Copyright 1998 Sokkia May-29-80 23:39 01/29/1999
                Angle Degrees        Dist Feet
              Temp Farenht         Coord N-E-Elev
JOB              TRAV                 Point Id Alpha (14)
            Atmos crn No         C and R crn No
          Record elev Yes      Sea level crn No
POS TP 1       North 10050.000      East 10000.000        Elev 0.000
POS TP 2       North 10000.000      East 10000.000        Elev 0.000
POS TP 3       North 9515.636       East 9551.975         Elev 37.611
              Code T3
POS TP 403   North 4967.527       East 5074.632         Elev 0.000
NOTE TS        Jan-01-80 00:14
** End of report **
```

Sokkia G2 This routine takes you directly to the SurvCOM dialog, similar to the Carlson SurvCE process.

TDS

Download [HP-48 and Husky]
In the TDS program, go to the File Transfer routine. Choose the type of data to transfer (CRD or RAW). Next pick the Send function key. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Download [Ranger and Windows CE]
In the TDS program, go to the Transfer routine and pick the Send File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. In the Type field of the file selection dialog, choose Coordinate Files or Raw Files. Stop here on the TDS and go to Carlson to run *Data Collection* in the Survey menu and pick TDS. Make sure that the COM port and baud rate are set correctly. Then pick the Download button. The Carlson program will now wait to receive the TDS file. Within 10 seconds select the file to send on the TDS and pick OK in the TDS dialog. The file should be transferred now. When the transfer is complete, the program will ask you for the Carlson file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.



Upload [HP-48 and Husky]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the File Transfer routine. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points but before clicking the Start Transfer button go to TDS and hit the Receive function key. Within 10 seconds go back and click the OK button on the range of points. The file should then transfer.

Upload [Ranger and Windows CE]

A Carlson coordinate (.CRD) file can be converted to a CR5 file and uploaded into TDS. Start in the TDS program, by going to the Transfer routine and pick the Receive File function. Set the "Connecting To" field to HP-48. Make sure that the COM port, baud rate and parity are set correctly and then pick OK. Then move back to Carlson and run *Data Collection* in the Survey menu and pick TDS. In the Carlson dialog, enter a TDS File name. This name should not include the drive and directory path or file extension. For example, if the coordinate (.CRD) file is c:\scadxml\data\simo2.crd then the TDS File name could be just SIMO2. Next pick the Select File button next to the Carlson coordinate (.CRD) file edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Now pick the Carlson Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the range of points and click the Start Transfer button.

SMI

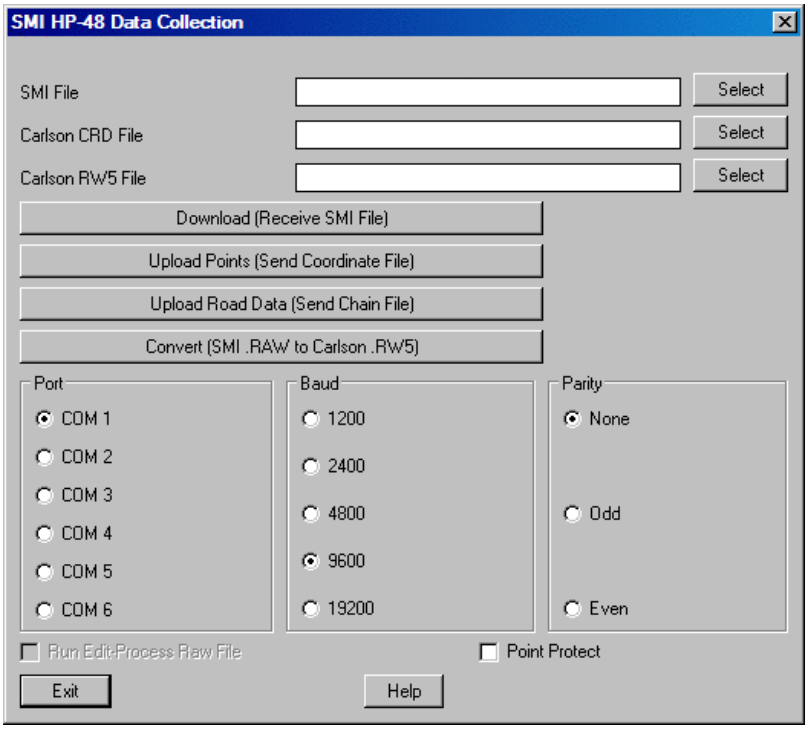
Download

To send point data from the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TOPC] [COMM]. In SMI version 6 or later, type [Job][KERM][SEND]. Also in version 6, make sure that the first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. The [NE] stands for North-East coordinate order which is the format that Carlson expects. Also in the [Job][KERM] screen, make sure that the second function key reads [COMM] and not [SPACE]. The [COMM] stands for comma separators. Then enter the first point to send followed by the last point to send but before pressing Enter

for the last point go to Carlson. Run *Data Collection* in the Survey menu and choose SMI. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to SMI and press Enter for the last point to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. To send raw data, use the [Print][Raw] routine in SMI along with the same Carlson procedure used for point data.

Upload

From the SMI data collector, go to the file transfer routine by typing [More] [NXT] [TO48] [COMM]. In SMI version 6 or later, type [Job][KERM][RECV]. Also in version 6, make sure that first function key reads [NE] and not [XY] in the [Job][KERM] screen. Otherwise the coordinate northing and easting will be reversed. Then enter the first point to send followed by the last point to send. Next enter the job name but before pressing Enter go to Carlson and run SMI under *Data Collection* in the Survey menu. In the Carlson dialog, specify the same job name as entered in SMI. Next pick the Select File button next to the Carlson CRD File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the same range of points as entered on the SMI. Go back to SMI and hit Enter for job name followed by clicking the OK button for range of points in Carlson. The file transfer should now go.



Leica

There are two types of Leica transfers: GIF-10 and GeoCom for all other Leica instruments. The type is set in the Equipment Type field on the main dialog. For transferring with the Leica instruments, the GeoCom program shows a dialog of the available COM ports on your computer. On the first time that you transfer to an instrument, you will need to pick the Instruments button and register the instrument from the list. Pick the Port Settings button to make sure that the communication settings match the instrument.

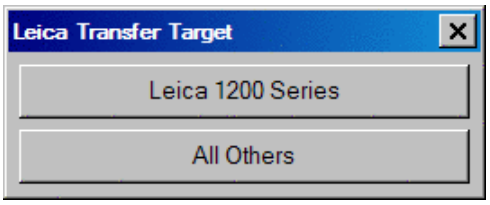
To download a file with GeoCom, make sure that the instrument is ON and connected to the computer by serial

cable. The instrument also needs to be in GeoCom mode. Then pick the Download in the Carlson dialog. In the GeoCom program, open the computer COM port that the instrument is connected to by picking the '+'. Then open the Memory Card and GSI folders. Next select the file to transfer and click the OK button. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

To upload a file with GeoCom, specify the file name to be created on the instrument in the Leica File field and pick the Upload button in the Carlson dialog. Then the program will prompt for the range of points to transfer. Fill out the range and pick the Start Transfer button. Then the GeoCom program will start. Open the computer COM port by picking the '+'. Then open the Memory Card folder and highlight the GSI folder and click OK.

The upload and download file transfer works with the GIF-10 data collector. The GIF-10 communication settings should be the following:

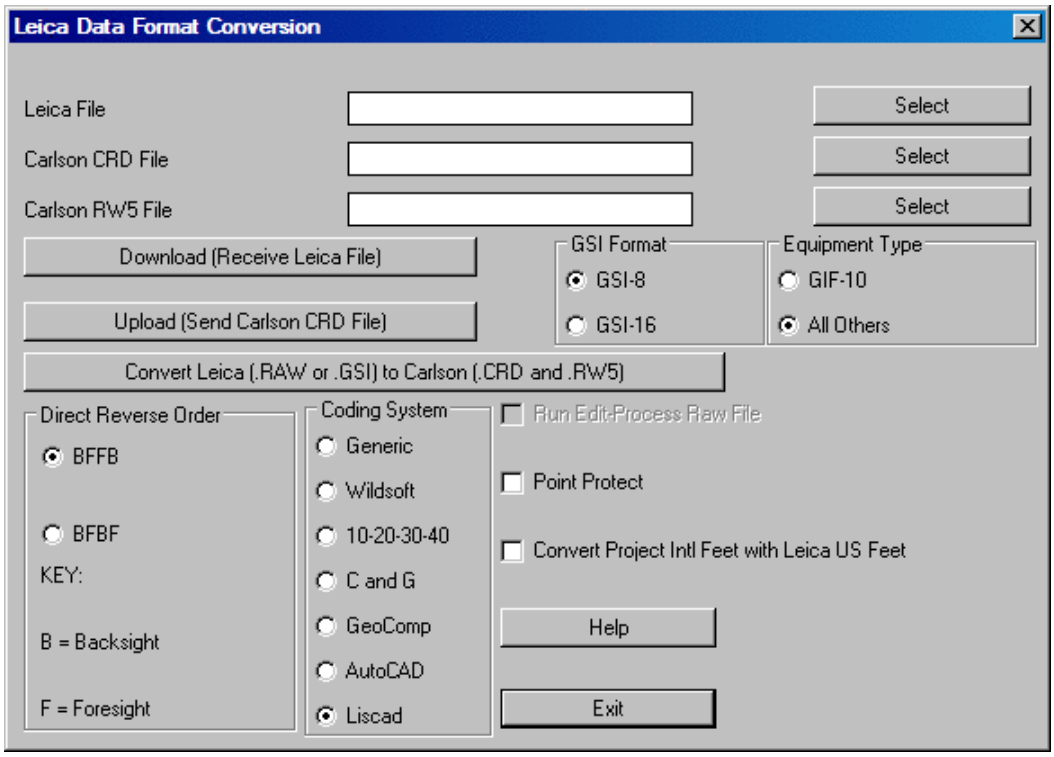
Baud: 9600
Parity: NONE
Protocol: NONE
Stop Bit: 1
End Mark: CR/LF
Connected As: Some computers use DCE and others use DTE



First Leica dialog



When Leica 1200 Series is chosen



All others

Download

From the GIF-10, go to the file transfer routine. Then go to Carlson and run *Data Collection* in the Survey menu and choose Leica. Check that the COM port and baud rate are set correctly. Then click the Download button and within 10 seconds go back to GIF-10 and select the file to send. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate (.CRD) file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

From the GIF-10 data collector, go to the file transfer routine. Then go to Carlson and run Leica under *Data Collection* in the Survey menu. In the Carlson dialog, specify the job name in the Leica File field. Next pick the Select File button next to the Carlson coordinate (.CRD) File edit box and choose the coordinate (.CRD) file to send. Check that the COM port and baud rate are set correctly. Then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Before clicking the OK button for range of points, go to GIF-10 and start the receive by highlighting Receive and pressing the Run button. The GIF-10 now shows the available job numbers. Choose a job to receive the transfer using the arrow buttons and then press the Run button.

Converting

Carlson supports raw and coordinate data collected using three different Leica Operation Codes: Wildsoft and 10-20-30-40 as well as the newer LISCAD. Moreover, data could be in the GSI8 format or the newer GSI16 format. Some example files are shown here.

GSI8 format data file using LISCAD Operation codes:

WILD GIF-12
410149+00000001 42....+00005003 43....+00005.42 44....+00005.25 45....+00005000
110150+00005000 21.324+35959480 22.324+09238590 31..01+00228271

410151+00000005 42....+00010100
110152+00005001 21.324+35156390 22.324+09303500 31..01+00133532
410153+00000005 42....+00070100
410154+00000014 42....+00000ELM
110155+00007082 21.324+34739450 22.324+09322050 31..01+00137685
410156+00000005 42....+00070102

GSI16 format data file using LISCAD Operation codes:

*110001+0000000000000001 84..11+0000010000000000 85..11+0000003000000000
86..11+0000000001000000 87..11+0000000000005170
*410002+0000000000000009 42....+0000000000000001 43....+000010000000.000 44....+000003000000.000
45....+000000001000.000
*410003+0000000000000001 42....+0000000000000001 43....+000000000005.330 44....+000000000000.000
*410004+0000000000000004 42....+00000000178.1530
*410005+0000000000000003 42....+0000000000000002 43....+0000000000000001
*110006+00000000000000RO 21.324+0000000017815300 22.324+0000000008424260
31..01+0000000000000000
*410007+0000000000000100
*410008+0000000000000012 42....+000000000005.090
*110009+0000000000000002 21.324+0000000000831230 22.324+0000000008130270
31..01+0000000000089996
*110010+0000000000000002 21.324+0000000018831230 22.324+0000000027829250
31..01+0000000000089996
*110011+00000000000000RO 21.324+0000000035815170 22.324+0000000027539300
31..01+0000000000000000

GSI8 format data file using Wildsoft Operation codes:

410001+00000001 42....+00000013 43....+00000000 44....+00000012 45....+00981101
410002+00000002 42....+00000013 43....+00005.42 44....+00000012 45....+00000000
410003+00000032 42....+00000500 43....+00004.26 44....+00000012 45....+00000000
410004+000000TP 42....+00000000 43....+00000000 44....+00000000 45....+00000000
110005+00000501 21.124+00000000 22.104+09136260 31...1+00000000 51..0.+0012+000
110006+00000502 21.124+03741320 22.104+08915570 31...1+00246818 51..0.+0012+000
110007+00000503 21.124+03915180 22.104+08919040 31...1+00251956 51..0.+0012+000
110008+00000504 21.124+06530420 22.104+08839360 31...1+00113998 51..0.+0012+000

Leica raw files usually have a .RAW or .GSI extension. The primary difference in the GSI8 and GSI16 formats is that information is contained in data blocks of 16 characters in the GSI16 format, while it is contained in blocks of 8 characters in the GSI8 format. Leica instruments make it possible to have both the GSI8 as well as GSI16 data formats in the same raw file. However, lines with the GSI16 format data will always start with an asterisk (*) character, to distinguish them from the GSI8 format. There is no distinction between Leica raw files collected in the Wildsoft and LISCAD operation codes.

Supported LISCAD codes:

1: New instrument setup
2: New target height
3: Sets of directions
4: Fixed azimuth
5: Feature code
6: Measured offset
8: Line creation for sub-codes 1 (straight string), 2 (curved string) and 6 (arc by 3 points)

9: Fixed coordinates
11: Close string
14: Additional description
20: Start of job
27: Feature code
90: Split feature code
100+: Descriptions

The Convert button can be used to convert any Leica format file into a Carlson format file. For example, if you have a Leica PCMCIA card then there is no serial cable transfer to do. Instead use the Convert routine to make the Carlson raw data (.RW5) and coordinate (.CRD) files. Since there is no distinction between Wildsoft and LISCAD files, the user must know in advance which format has been used in the file. Then, select that particular option (Wildsoft, 10-20-30-40 or LISCAD) under the "Coding System" option at the bottom of the dialog box, as shown in the previous page. Another option that the user needs to choose is the order in which foresight-backsight readings have been recorded in the raw file, BFFB or BFBF, as explained in the dialog box. Then, the user can simply pick the "Convert" button and the program prompts for the input" Wild/Leica File" (raw file), and the output" Carlson RW5 file" and "Carlson CRD file", if they are not already filled.

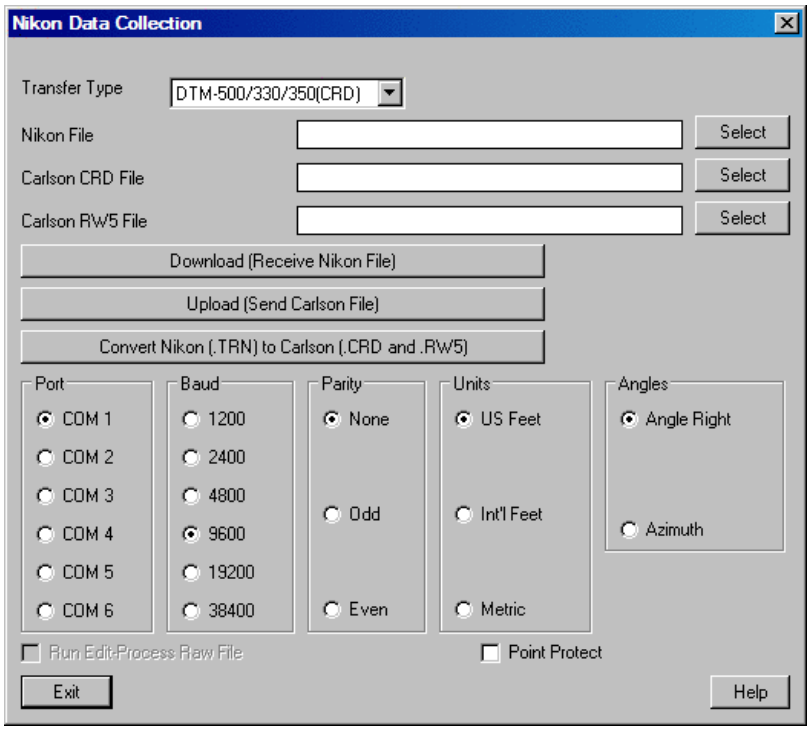
Nikon

Download

First choose the equipment and data type under the Transfer Type list. Also check that the communication and data format settings match your collector. Then click the Download button and follow the on-screen directions. When the transfer is complete, the program will ask you for the Carlson coordinate file (.CRD) and raw file (.RW5) to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector. The original data from the collector is stored in a file name with the same name as the coordinate file except with a .TRN extension. For example, job5.crd would have job5.trn.

Upload

Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Set the points and then click the Start Transfer button. The file transfer should now go.



Convert Nikon to Carlson

The Convert button will translate the Nikon raw file format (.TRN or .RAW) into Carlson coordinate (.CRD) and raw (.RW5) files.

Portion of typical Nikon file format:

MP,1,,5000.0000,5000.0000,0.0000,T/1
CO,31-Oct-1999 11:42:38
ST,1,,2,,0.0000,0.00000,0.00000
SS,3,0.0000,152.1510,359.59590,90.44100,11:43:38,T/2
SS,4,0.0000,127.5560,0.06040,90.40110,11:44:45,CON
SS,5,0.0000,97.1820,2.19580,90.52460,11:45:43,CON

Geodimeter

Download

From the Geodimeter data collector, go to the file transfer routine by pressing the PRG (Program) key and entering program 54. Then choose Imem (option 1) as the source. Next choose the file type to send as either Job (measurement data) or Area (point data). The Geodimeter will then prompt for the job name. Next enter Serial (option 3) as the destination. A confirmation screen appears showing the serial port settings. Here are some typical settings:

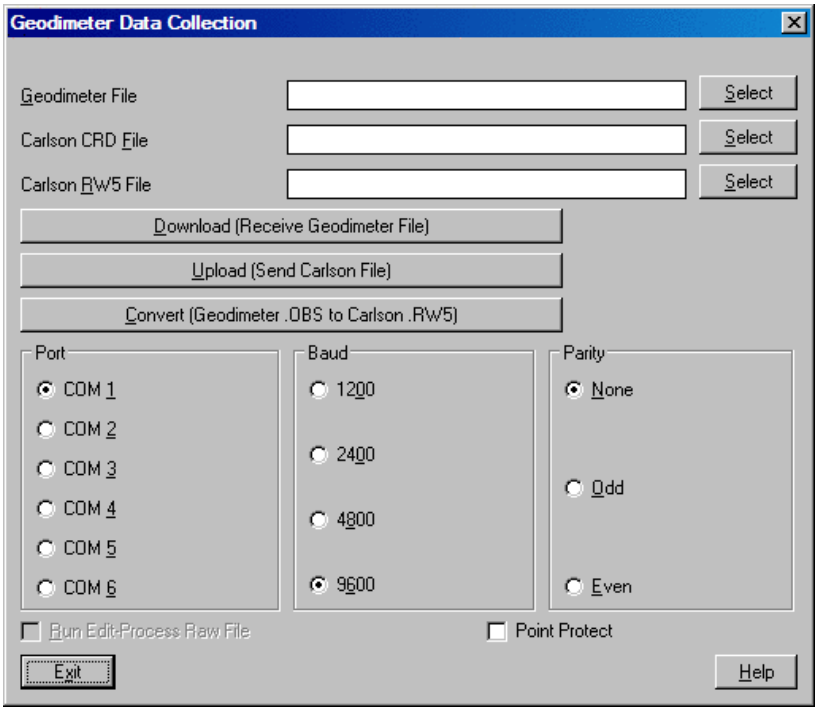
COM=1,8,0,9600

Before pressing enter (ENT key), go to Carlson and run *Data Collection* in the Survey menu and choose Geodimeter. Then click the Download button and within 15 seconds, go back to the Geodimeter and press Enter. The file transfer should now go. When the transfer is complete, the program will ask you for the Carlson coordinate file and raw file to create if you haven't already specified a file name in the dialog. With Point Protect on, the routine will check the coordinate file for existing point data before downloading the point from the data collector.

Upload

In Carlson, run Geodimeter under *Data Collection* in the Survey menu. Pick the Select File button next to the Carlson CRD File edit box and choose the CRD file to send. Check that the COM port and baud rate are set correctly and

then click the Upload button. A dialog now allows you to specify the range of point numbers to upload. Enter the points to send but before clicking OK, go to the Geodimeter data collector. Start the file transfer routine by pressing the PRG key and entering program 54. Then choose Serial (option 3) as the source. The Geodimeter will display the serial port settings. Check these values and press enter. Next choose Area (option 2) as the destination. Then enter the job name. The Geodimeter is now listening for data. Quickly go back to Carlson and click OK on the points to send dialog. The file transfer should now go



Convert

The Convert button will translate the Geodimeter raw file format (.OBS) into Carlson coordinate (.CRD) and raw (.RW5) files.

Communication Settings

If the Geodimeter is not communicating with Carlson, run function 79 on the Geodimeter and make sure that it is set to 4. This setting is for the transfer message end of sequence format.

Supported Geodimeter Codes

The following Geodimeter codes are processed when converting the Geodimeter raw file. All other codes are recorded as descriptions in the Carlson rw5 file.

0=Info
1=Data
2=Station No
3=Instrument Height
4=Point Code
5=Point Number
6=Signal Height
7=Horizontal Angle
8=Vertical Angle
9=Slope Distance
11=Horizontal Distance
17=Horizontal Angle

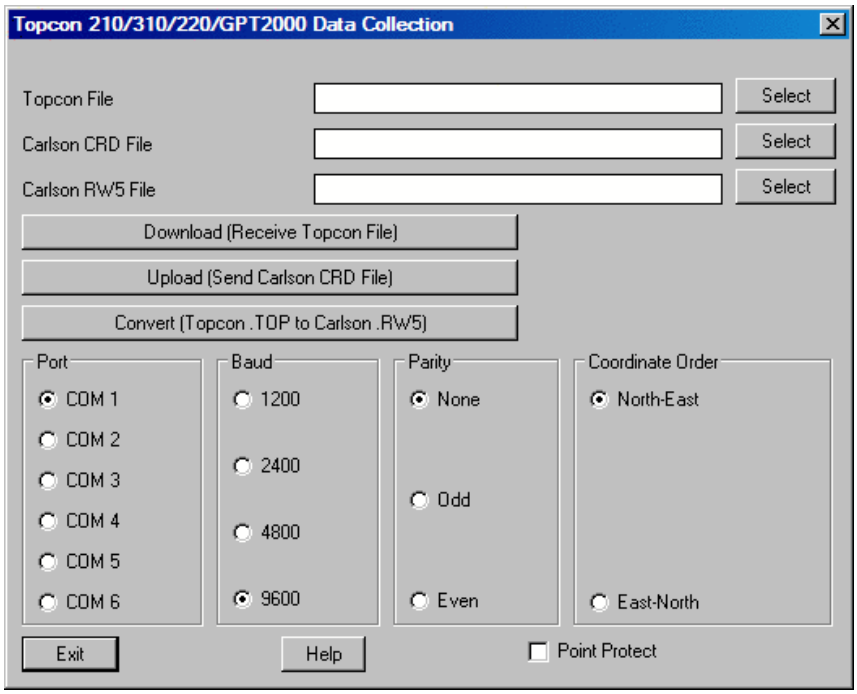
18=Vertical Angle
21=Horizontal Reference Angle
30=Atmospheric Correction
37=Northing
38=Easting
39=Elevation
40=Delta North
41=Delta East
42=Delta Elevation
45=Correction To Bearing
46=Standard Deviation
50=Job Number
51=Date
52=Time
53=Operator
54=Project Id
55=Instrument Id
56=Temperature
60=Shot Id
61=Activity Code
62=Reference Object
70=Entered Radial Offset
71=Entered Angle Offset
72=Calculated Radial Offset
73=Calculated Angle Offset
74=Air Pressure

Portion of typical Geodimeter file format

5=108
4=13POC
6=5.000
7=238.0708
8=89.2236
9=440.39
37=767.42
38=4626.07
39=699.795

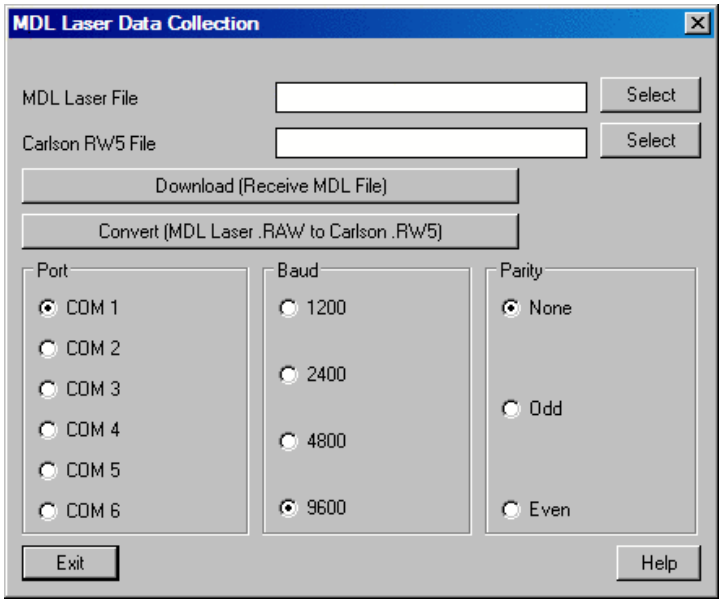
Topcon 210/310/220/GPT2000

This command supports these above Topcon models.



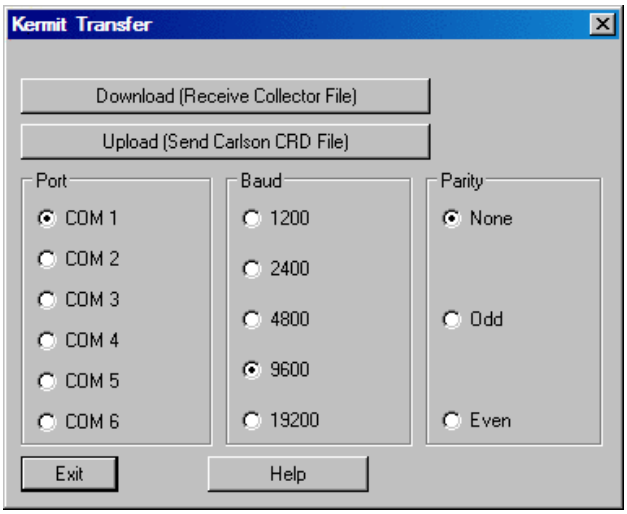
MDL Laser

The MDL Laser outputs a raw file of angles, distances and codes as one long string of data which can be converted into a Carlson raw data (.RW5) file. There is no coordinate data in the MDL raw file. So you need to run *Edit-Process Raw File* to calculate coordinates from the raw data. The Download button will transfer the MDL raw data from a BDI logger.



Kermit

Kermit can be also used for transfering files with accuracy. The dialog looks like this:



Keyboard Command: datacolt
Prerequisite: None

Convert LDD-AEC Contours

Function

This command allows you to convert LandDesktop contours (known as AECC_CONTOUR objects) into polylines. You must have the AEC Object Enabler installed before using this command. If you do not have the object enabler installed, download the latest version from www.autodesk.com.

You can use the *List* command to determine if contours are polylines or AECC_Contour objects. Here is an example listing:

AECC_CONTOUR Layer: "CONT-MJR"

Space: Model space

Handle = 429

Major Contour Interval

Elevation: 1005.00

Smoothing: None

Number of Vertices: 48

Open

Length: 560.25

Constant width: 0.00

Style Name: Standard

Import/Export LandXML Files

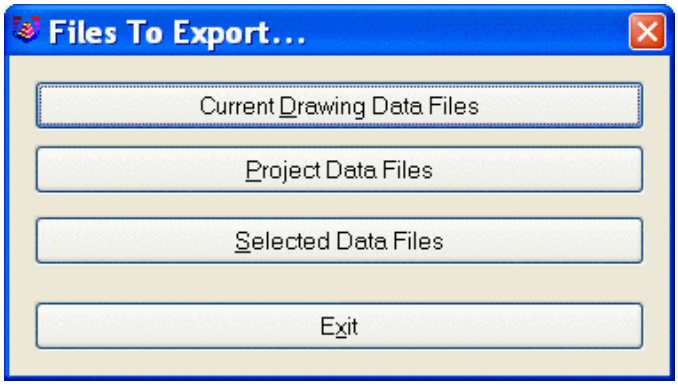
Function

This command performs two functions: Export Carlson files to LandXML and Import LandXML files into Carlson. This command supports version 1.0 of LandXML and the following Carlson file types: Coordinate (.CRD) files, Centerline (.CL) files, Profile (.PRO) files, Section (.SCT) files, Grid (.GRD) files, Triangulation (.FLT) files, and Lot (.LOT) files.

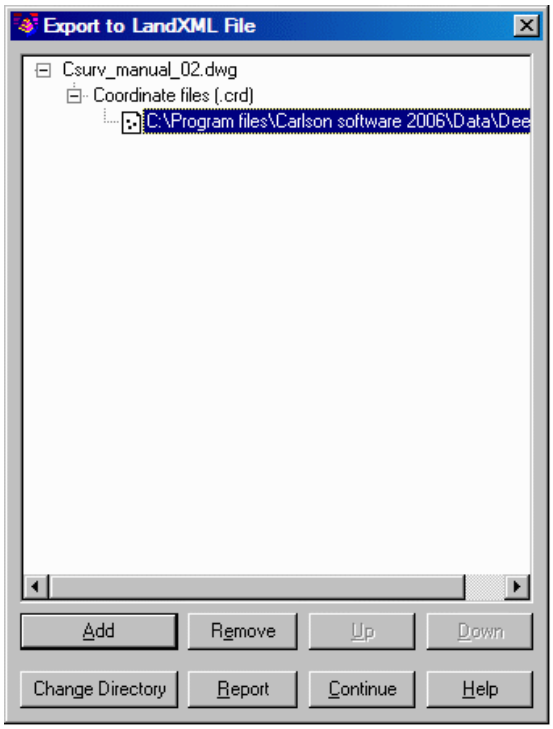


Export to LandXML: Allows you to export a Carlson file to LandXML. Choose a new or existing LandXML file, then choose an existing Carlson file.
Import to Carlson: Allows you to import a LandXML file into Carlson. First, choose a new or existing Carlson file, then choose an existing LandXML file.

You first see the Import/Export LandXML dialog. To Export, for example, first click Export to LandXML. Then click Current Drawing Data Files button. You will then see the Select LandXML File dialog box. Select an existing .XML file to be exported from the list of files on the right and click Open.



The next dialog appears, showing the new buttons to be used for the Export. Choose the Add, Remove, Change Directory and/or Report buttons, and then Continue.



Choosing the Continue button takes you further into the Export process. The Report button will give you the Report Formatter Options dialog box.

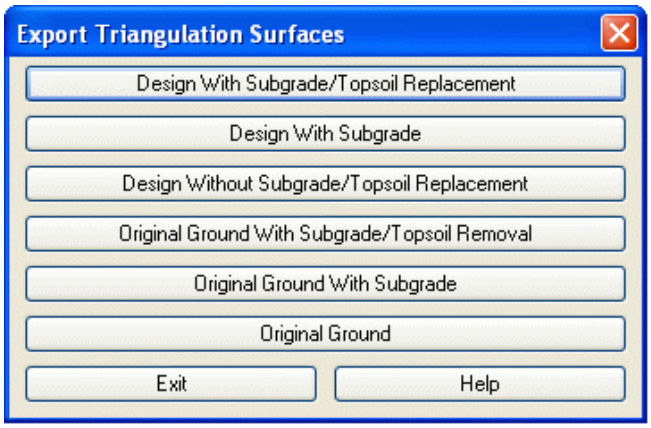
Data protection is turned on by default, meaning that if you are importing/exporting to an existing file, you will be prompted before the program overwrites existing data. There is a Point Protection option that will not allow any points to be overwritten if the same number appears more than once in the XML file being imported. If you decide to Import From LandXML, go back to the opening dialog an choose the existing LandXML file.

Keyboard Command: imp_exp
Prerequisite: Files to convert

Import/Export Carlson Triangulation Files

Function

Import Carlson Triangulation Files allows you to import an external surface file into TakeOff to use as a named surface. Export Carlson Triangulation Files allows you to take a Surface Triangulation file and save it independent of the drawing.



Prerequisite: .TIN or .FLT files

Keyboard Command: import_tin, export_tin

Import/Export DXF Files

Function

Import DXF File allows you to import an external DXF file into the Carlson TakeOff drawing (DWG). Export DXF File exports selected entities from the Carlson TakeOff drawing (DWG) to a DXF file. The DXF file is another format for the drawing objects of the DWG file. Typically, other CAD programs at least support DXF files if they don't support DWG. So the DXF file (Drawing eXchange File) can be used to transfer drawing data between Carlson Takeoff and other non-DWG programs.

Prerequisite: a DXF file

Keyboard Command: _dxfin, _dxfout

Import Polyline File

Function

This command draws polylines from the selected polyline file. These polylines are drawn in the current layer. This command supports the following formats: Carlson (.PLN), Idan (.DIS), MicroStation (.TXT), MOSS (.INP, .PRN) and Topcon Pocket 3D (.TXT).

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation/MOSS/Topcon]? *press Enter to accept Carlson*
Polyline File to Read Dialog *select existing .PLN file*

Keyboard Command: polydraw
Prerequisite: A polyline file

Export Polyline File

Function

This command creates a polyline file that contains the point data of the select polylines. The objects supported by this tool include polylines, arcs and lines. If you want to include text, you must use the *Text Explode To Polylines* command found in the Edit menu to convert the text to polylines before running this command. This polyline file is a text file that has three formats. The Carlson format (.PLN) is used by machine control (Carlson Grade, Dozer 2000, GradeStar) for the plan view. Each polyline begins with a line of "POLYLINE, Color number". Then the points for the polyline are listed on separate lines in X,Y,Z format. Here is a list of the available color numbers:

0 = Black
1 = Blue
2 = Green
3 = Cyan
4 = Red
5 = Magenta
6 = Brown
7 = Light Gray
8 = Dark Gray
9 = Light Blue
10 = Light Green
11 = Light Cyan
12 = Light Red
13 = Light Magenta
14 = Yellow
15 = White

The MicroStation format (.txt) can be imported into MicroStation. This format has the coordinates as space delimited for each polyline point. There is an extra column with a 1 or 0 where 1 specifies the start of a new polyline. The DTM and Idan formats create linework files for the DTM and Idan programs.

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation]? *press Enter for Carlson format*
Specify File to Write dialog *create a new file or append to existing*
Polyline file for Grid File Utilities macro [Yes/<No>]? *press Enter* The option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.
Include Z coordinate in polyline file [Yes/<No>]? *press Enter* This option controls whether the polyline vertices are written in 2D or 3D.
Specify Exclusion/Warning Polylines [Yes/<No>]? *press Enter* This option applies to machine control for warning areas.
Specify WorkZone Polylines [Yes/<No>]? *press Enter* This option applies to machine control for working areas.
Reduce Polyline Vertices [<Yes>/No]? *press Enter* This option applies Reduce Polyline to the polyline vertices before writing the file.
Enter reduce offset cutoff <0.1>: *press Enter*
Decimal places for coordinates <2>: *press Enter*
Select polylines, lines and arcs to write.
Select objects: *pick the entities to process*
Done.

Sample Polyline File:

POLYLINE,15
47639.82,74540.11,0.00
47670.49,74565.79,0.00
47701.08,74591.49,0.00
49375.61,76358.47,0.00

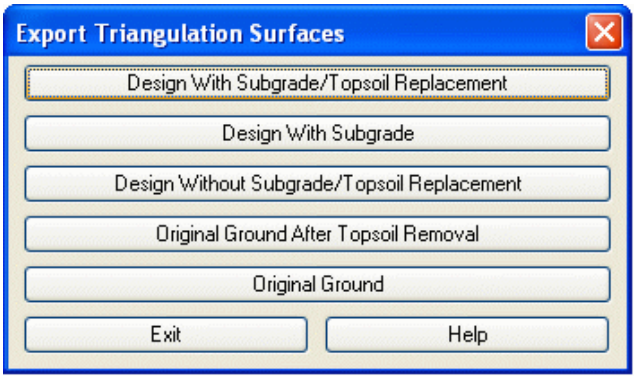
50066.86,76846.75,0.00
POLYLINE,15
47633.24,74547.97,0.00
47663.90,74573.65,0.00
etc...

Keyboard Command: polywrite
Prerequisite: Polylines in the drawing

Export Surface DXF Files

Function

Export Carlson Triangulation Files allows you to take a Surface file and save it as a DXF file. The DXF file will contain 3D Faces for the triangulation surface. This command is another way besides the LandXML routine to transfer a Carlson Takeoff surface to other programs.



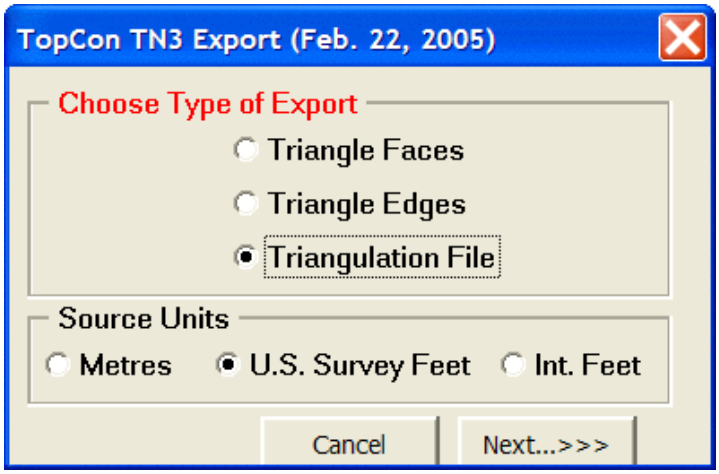
Prerequisite: .TIN or .FLT files

Keyboard Command: EXPORT_DXF

Export Topcon Tin File

Function

This command writes a Topcon TIN file (.TN3) from a Carlson Takeoff surface. The first prompt at the Command line chooses whether to export the surface from a file or the screen. The file option will prompt for a triangulation file (.tin or .flt) and then the Topcon TIN file to create. The screen option can be used to export a surface from 3d Faces or TIN lines from the drawing. The screen option has the following prompts:



•

Choose Type of Export

- **Triangle Faces:** Triangulation networks that consist of edge matched 3DFaces defined by three points.
- **Triangle Edges:** Triangulation Networks that consist of edges drawn with Lines defined by two points.
- **Triangulation File:** A Carlson triangulation file (with a DOS extension of .flt).

Source Units: Select the appropriate unit type of the entities to be exported, then press NEXT.

If the Triangulation File option is chosen, a standard windows file selection dialog will open to allow for the selection of the .flt file.

If Triangulation Edges or Faces are used for the export, you must select the entities to be exported from the screen. The following dialog opens:



Manually Select allows on-screen selection with any of the various methods (Window, Crossing, Crossing Polygon, etc.)

All on a Selected Layer allows selection of a single entity. The command then selects all Lines or 3DFaces on the layer of the selected entity forexport, filtering entities not on selected layer.

Generally, you can select any entities drawn on the screen without special care. The command filters out all types except Lines or 3DFaces depending on the type selected.

When Processing Lines: Crossings, crossing polygon, and fence selection sets can often result in "spikes" around the edges. The command generally discards these spikes, or reconnects them. In any case, better results are developed with clean selections of drawing entities.

Pressing NEXT will open the following dialog:

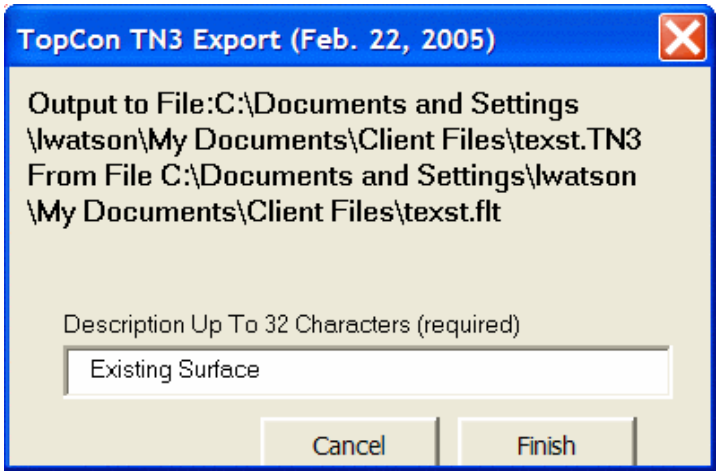


Other Features to Export: Often times, landmark lines, or other features will assist in orientation to the user when using the TN3 file in the field. For example: Property lines, project centerlines, or other well established reference lines. These features may be added and displayed on the TN3 by choosing them at this prompting. The Other Features dialog opens with all three Source Formats.

Selections are made as described above.

Once the selections are made and the dialog reappears, press NEXT.

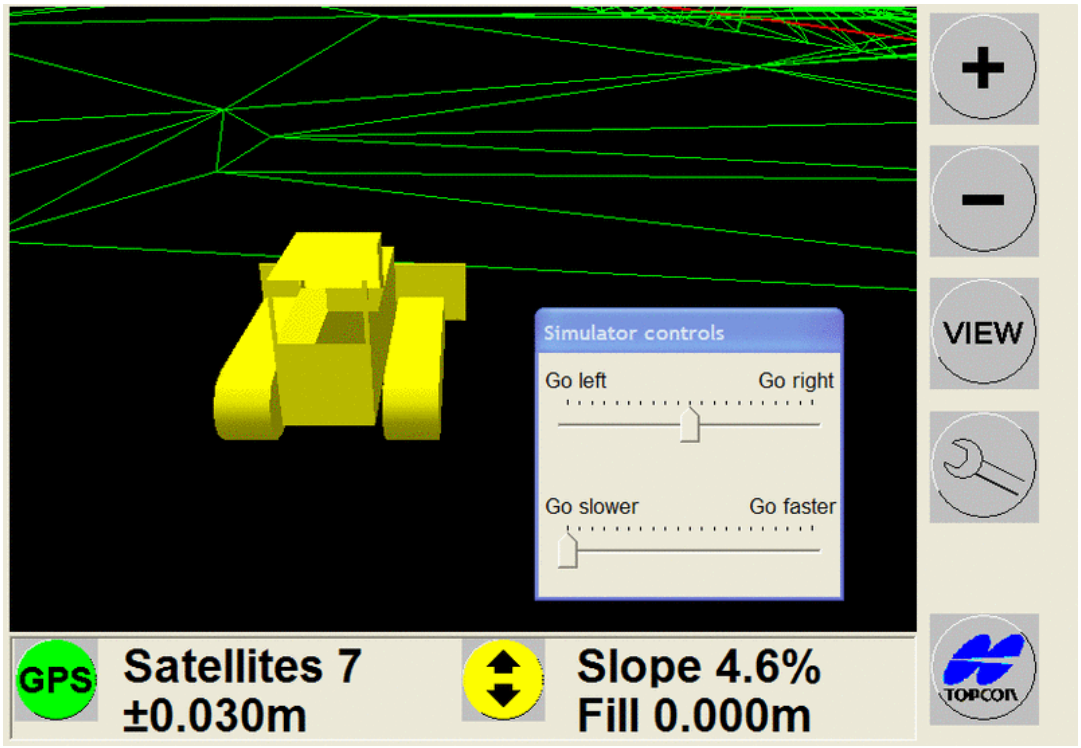
Naming the TN3 File: After all the triangulation and feature items are selected from the graphics screen, the command requests a name for the output file. Once a name is given, the following dialog reviews all the selections made. A description for the job can be added at this time. Press Finish when ready to proceed.



The program will process the selected entities and/or files and report as shown here:



To view the final output, press the Run Simulation button, or press Exit to return to the Carlson program.



Run Simulation Dialog

Clipboard

Function

This command allows for different cut, copy, and paste options.

Cut

To cut objects to the Clipboard.

- Select the objects you want to cut.
- From the Clipboard command, choose Cut.

The objects are available to be pasted into other Windows applications.

Copy

To copy objects to the Clipboard.

- Select the objects you want to copy.
- From the Clipboard command, choose Copy.

Copy with Base Point

To copy objects to the Clipboard. When the objects are pasted into a drawing, the program places them relative to the specified base point.

- Select the objects you want to copy.
- From the Clipboard command, choose Copy with Base Point.
- Specify the base point.

Paste

The objects currently on the Clipboard are pasted into the drawing at the specified insertion point.

- From Clipboard command, choose Paste.

Paste as Block

The objects currently on the Clipboard are pasted into the drawing as a block at the specified insertion point.

- From Clipboard command, choose Paste as Block.

Paste to Original Coordinates

The objects currently on the Clipboard are pasted into the drawing using the coordinates from the original drawing.

- From Clipboard command, choose Paste to Original Coordinates.

Display Last Report

Function

This brings up the last report generated by any Carlson Takeoff command that uses the standard report viewer.

Prerequisite: a previously generated report

Keyboard Command: last_report

Exit

Function

This command allows you to exit Carlson TakeOff.

This option allows you to quit the program if there have been no changes since the drawing was last saved. If the drawing has been modified, the program displays the Drawing Modification dialog box to prompt you to save or discard the changes before quitting.

You can close a file that has been opened in read-only mode if you have made no modifications or if you are willing to discard them. To save modifications to a read-only drawing, use the SAVE AS command under the File menu.

Prerequisite: None

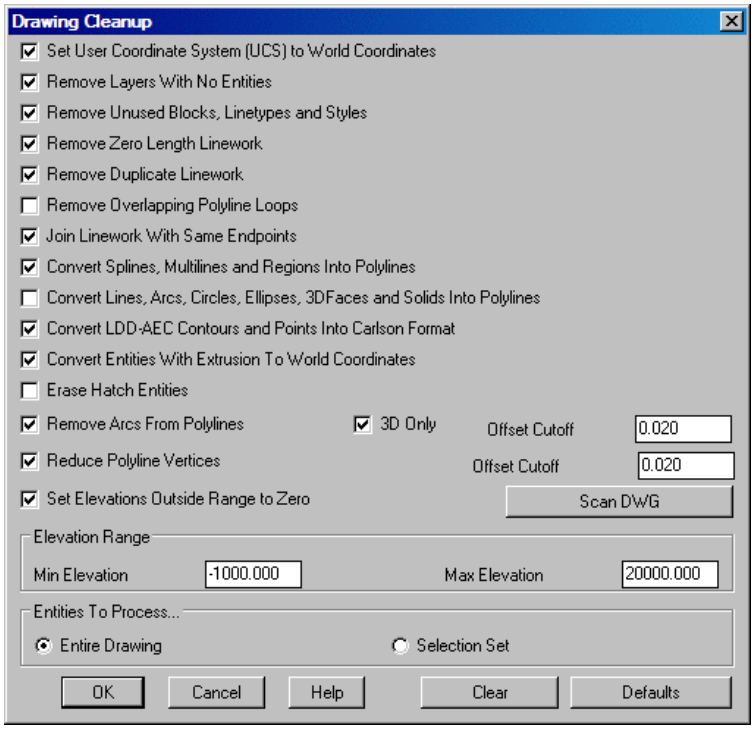
Keyboard Command: QUIT

Tools Menu 4

Drawing Cleanup

Function

The Drawing Cleanup dialog box allows you to perform many functions that fix common errors, and it removes unnecessary data found in many drawing files. It also converts incompatible data into useful entities. This command offers many filters that audit the drawing file and allows you to select which options and settings you want to use. A report of the cleanup results will be displayed upon completion. Always save your file when the drawing cleanup routine is complete.



Set UCS to World Coordinates
This sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson works exclusively in the world coordinate system and there is no way to change this setting. In AutoCAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, click this on to restore the UCS.

Remove Layers With No Entities
AutoCAD drawings work with a "BYLAYER" concept meaning that layer definitions define the drawing. For example, the layer named EOP might be used to display polylines at the Edge Of Pavement in the drawing. Many times extra layers get defined by a user but not used to display any objects. This function removes any layers defined in the drawing that are not being used.

Remove Unused Blocks, Linetypes and Styles
This functions removes this unused information from the drawing.

Remove Zero Length Linework
This function seeks out and removes any linework definition that have zero length. Point nodes are not removed.

Remove Duplicate Linework
This function finds any duplicate linework in the drawing and removes all but one set.

Remove Overlapping Polyline Loops
Polylines that completely overlap themselves are broken into two different polylines.

Join Linework With Same Endpoint
This function finds common endpoints on linework on common layers with common elevations and joins the linework into a continuous polylines. This is very helpful for future selection sets.

Convert Splines, Multilines and Regions Into Polylines
Some CAD applications utilize Spline Object Definitions and Regions, Carlson utilizes basic polyline/polygon definitions. This function finds any Splines and/or Regions defined in the drawing and re-defines them as simple polylines or polygons.

Convert Lines, Arcs, Circles, Ellipses, 3DFaces and Solids Into Polylines
By converting Lines, Arcs, Circles, Ellipses, 3D Faces, and Solids into Polylines, you can use the variety of Polyline commands available in Carlson.

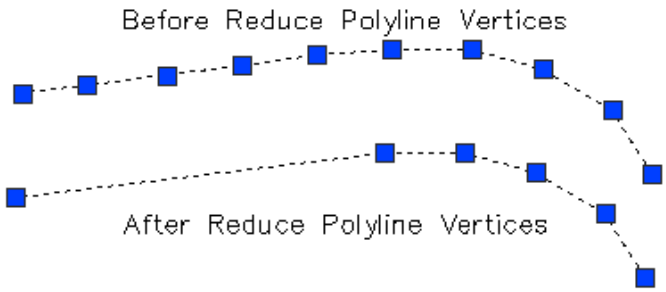
Convert LDD-AEC Contours and Points Into Carlson Format
Drawings created in the Land Development Desktop CAD program can contain special objects known as LDD-AEC contours that define their topographic contour display. This function locates those special objects and re-defines them as simple 2D polylines retaining their elevation values.

Convert Entities With Extrusion To World Coordinates
Drawings created in the Land Development Desktop CAD program can contain special objects known as LDD-AEC contours that define their topographic contour display. This function locates those special objects and re-defines them as simple 2D polylines retaining their elevation values.

Erase Hatch Entities
Carlson offers many hatch display options, however hatch entities have no 3D value. This function removes all hatch entities in the original drawing to help reduce the size and clutter of the drawing file.

Remove Arcs From Polylines - Offset Cutoff
This function replaces arcs in polylines with a series of short chord segments. The purpose is to prepare the polylines for modeling since arcs need to be converted into segments to be part of the triangulation model. The density of of chord segments is controlled by the offset cutoff. This cutoff represents how much the polyline can move horizontally. A smaller cutoff will result in more chord segments. The option for 3D Only controls whether only polylines at zero elevation or both zero and elevated polylines get processed. Sometimes you may want to leave the arcs in zero elevation polylines when these polylines represent road alignments and are not part of the surface model.

Reduce Polyline Vertices - Offset Cutoff
This function utilizes a pre determined offset amount and removes unnecessary polyline vertices that fall within the offset amount.

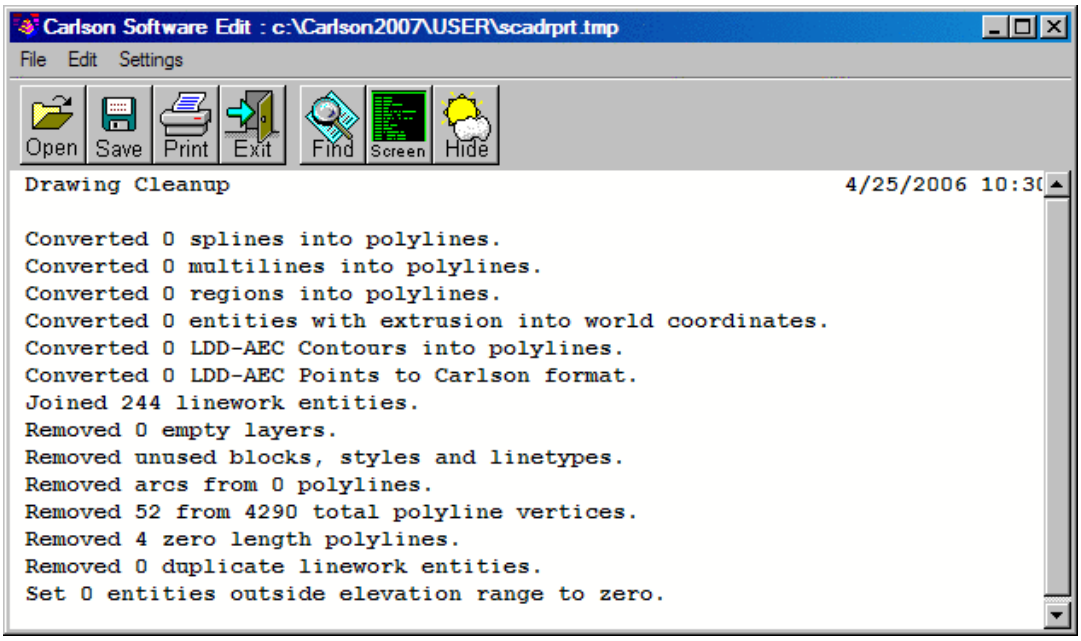


Set Elevations Outside Range to Zero and Elevation Range
This function comes with a "Scan DWG" option that audits the elevation range in the drawing file. Once the minimum and maximum elevation range has been set, manually or by a scan, all objects that fall outside the set range are moved to elevation zero. All objects at zero elevation do not contribute to the 3D model.

Entities To Process...
This allows you to run the command for the entire drawing or for a selected set.

Default
This allows you to return to the Carlson Drawing Cleanup default settings.

Final Report
This example report displays the results of drawing cleanup. Like all reports in Carlson, this report can be saved to a text file, sent directly to your printer, or pasted onto the screen ad text entities.

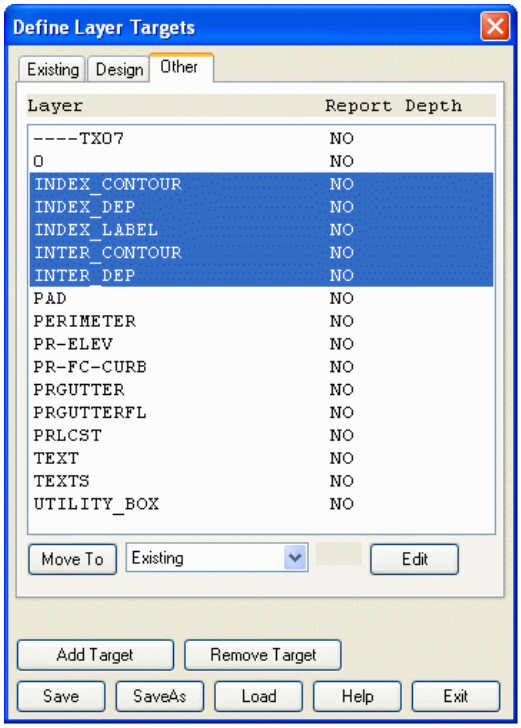

Carlson Software Edit : c:\Carlson2007\USER\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Drawing Cleanup 4/25/2006 10:3(
Converted 0 splines into polylines.
Converted 0 multilines into polylines.
Converted 0 regions into polylines.
Converted 0 entities with extrusion into world coordinates.
Converted 0 LDD-AEC Contours into polylines.
Converted 0 LDD-AEC Points to Carlson format.
Joined 244 linework entities.
Removed 0 empty layers.
Removed unused blocks, styles and linetypes.
Removed arcs from 0 polylines.
Removed 52 from 4290 total polyline vertices.
Removed 4 zero length polylines.
Removed 0 duplicate linework entities.
Set 0 entities outside elevation range to zero.

Keyboard Command: dwg_cleanup
Prerequisite: None

Define Layer Target/Material/Subgrade

Function

The Define Layer Targets dialog box (shown here) offers many functions that will ultimately make up the surface models used in volume and material calculations. Every entity (line, polyline, point, etc) in a drawing is assigned a layer name. Carlson Takeoff uses the entity layer names to define which entities represent the existing ground surface, the design surface or no surface. These surfaces are referred to as the "Target" surfaces.The drawing entities are assigned their target surface by their layer name. For example, if polylines representing design contours are on the layer "Final", then "Final" will be set as a layer for the design surface. For layers of entities that are for neither existing nor design surfaces (such as text labels for street names), the layer target is set to Other. The Define Layer Targets dialog has three lists for layer targets: Existing, Design and Other. To switch between lists, pick the tabs at the top of the dialog. To move a layer to a target destination, highlight the desired layer, choose the target from the Move To list and pick the "Move To" button. All layers populate the "Other" target before being assigned to "Existing" or "Design".



Besides the basic three layer targets (Existing, Design and Other), you can add more target surfaces with the Add Target button. When another target is defined, there will be another tab along the top of the Define Layer Targets dialog. Then layers can be assigned to this additional target surface. The only pre-defined additional surface is Overexcavate. The layers that are assigned to the Overexcavate target can be modeled into the Overexcavate surface using the Make Overexcavate Surface command. Besides Overexcavate, the other additional targets are user-defined. The layer targets can be modeled using the Make User-Defined Surface command. Then these

surfaces can be used in Takeoff commands by assigning them to a Takeoff existing or design surface using the Set Active Surfaces command.

Edit Materials

The "Edit" button activates the Edit Material dialog box (shown here) and allows you to define the Material name and Subgrade depths and names. Besides assigning target surfaces by layer, layers are also used to define material names and subgrades depths. By assigning a material name, Subgrade names and depths to layers, the volume, area, length and count for entities on these layers can be reported. Also the depth is used to vertically adjust the design surface. **The polylines on the layer used for a Materialmust be closedpolylines.** Carlson Takeoff supports nested Subgrade polylines for exclusion areas such as islands by counting how many Subgrade polylines surround an area. If the number is odd, then the area is included in the Subgrade. The even count regions in the area are not part of the Subgrade. To activate the Edit Material, select a layer from the list and then choose "Edit".

Include in Material Quantities Report

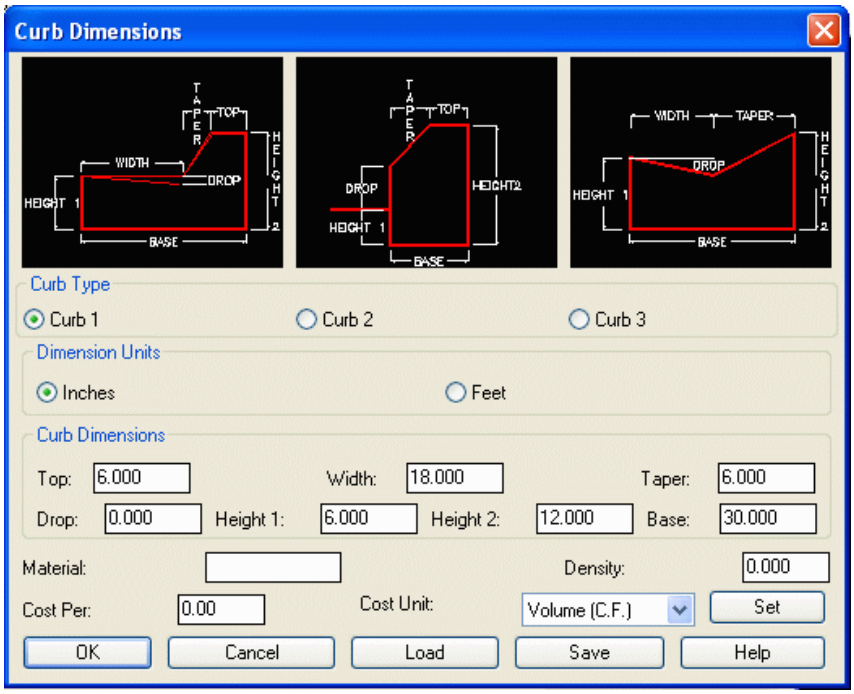
With this option checked on, the material that is named will appear in the Material Quantities Report. The report will include either the area of the material, the linear length of the material, or the number of items counted on the layer defining the material. This is accomplished by choosing "Area", "Linear", or "Count" for the Material Type.

Set Color For 3D Drive

This options checked allows you to assign a color for this particular material for display purposes during the 3D view/drive simulator.

Material Type

This will report the subgrade by area, linear length, count, or as curb area. If you choose Back of Curb/Pavement then you can pick on the Curb Dimensions button and bring you to the below dialog:



This option will calculate your curb volume as well as act as the limit of the pavement. The pavement limit will be from the Back of Curb polyline offset by the length of the Curb base. In the above case the base is 30 inches wide. Therefore, the pavement area will stop 30 inches before the Back of Curb polyline.

Material Cost Per Cost Unit

Use this field to add the value of the multiplier for the unit cost of your material. If the material type is an area that has multiple subgrades, use the available fields below to add each individual subgrade name, depth and cost value per unit type. If a linear or count type material type option is selected, use the "length in feet", or the "count" unit options.

Adjust Design Surface by Depth

This determines whether the subgrade depths are incorporated in the design surface or not.

Area Subgrades

Depth Units

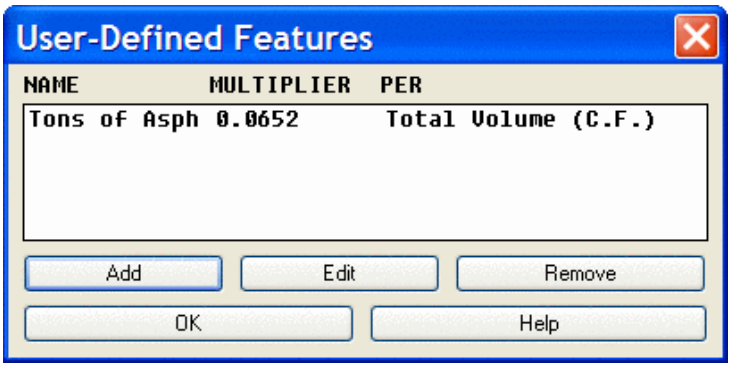
Select the "feet" or "inches" as the unit value desired for depth of subgrades.

Subgrade Name Depth Shrink Cost Per Cost Unit Density

Use these options for areas that are represented with a single/multiple closed polygon/polygons in the drawing, but have multiple material types defining the surface. Simply name each "lift" in the area, issue a depth value and add a cost unit if desired, or click on select and choose a material from the Materials Library (see Define Materials Library for more). Carlson Takeoff will report each subgrade material vaule in the material quantities report. The

Shrink factor is multiplied by the subgrade volume in the material quantities report and represents the fill shrinkage. A Density factor can be entered in when using Cost Per Tons.

If user entered values are needed in the report use the "Edit User-Fields" button to activate the "User Defined Features" dialog box shown here. Choose the "Add" button to define needed fields shuch as TONS of material or BAGS OF GRASS SEED for reporting options.



Once all of the material subgrades, depths and cost units or user defined units have been defined, select save to prserve the settings in a .trg file, the "save as" function allows the user to name the file to load later.

Prerequisite: none
Keyboard Command: define_tk_layers

Edit Selected Layer

Function

Use this command to click on any layer and advance to the Edit Materials dialog from the Define Layer Target/Material/Subgrade command.

Prerequisite: none
Keyboard Command: edit_tk_layer

Set Layer For Existing

Function

Set Layer For Existing allows the user to pick the layers from objects on the screen and assign them to the Existing Layer.

Prerequisite: none
Keyboard Command: set_existing_layer

Set Layer For Design

Function

Set Layer For Design allows the user to pick the layers from objects on the screen and assign them to the Design Layer.

Prerequisite: none
Keyboard Command: set_design_layer

Set Layer For Other

Function

Set Layer For Other allows the user to pick the layers from objects on the screen and assign them to the Other Layer.

Prerequisite: none
Keyboard Command: set_other_layer

Boundary Polyline

Function

The Boundary Polyline options allow the user to Set the Boundary Polyline, Set the Exclusion Polylines, Clear Exclusion Polylines, Hatch the Boundary Area, Erase the Boundary Hatched area.

Set Boundary Polyline

Use this command to select the "CLOSED" polyline that defines the outer most limit of the disturbed area. This boundary should dissect the site at the point where the design contours meet the existing contours, or where the limit of work will occur. If your site contains seperated areas (such as different phases or isolated sections of work), then multiple Boundry Polylines can be used. Volume calculation will take place inside this boundary.

Prerequisite: a closed polyline
Keyboard Command: tag_inclu

Set Exclusion Polylines

Use this command to select the "CLOSED" polylines the define the areas inside the Boundary Polyline that will not be disturbed. These boundarys should also be at the intersection of the proposed and existing surface. A pond or wetland that will not be removed during construction is a good example of an Exclusion Area.

Prerequisite: a closed polyline
Keyboard Command: tag_exclu

Clear Exclusion Polylines

Use this command to select polylines that were previously defined as exclusion polylines but are no longer needed as exclusion areas.

Prerequisite:
Keyboard Command: untag_exclu

Highlight Boundary Polylines

This command highlights the polyline you set as the Boundary Polyline.

Prerequisite: a boundry polyline
Keyboard Command: highlight_boundary

Hatch Boundary Area

Use this command to confirm the boundary polylines that have been selected are correct. This hatched area can also be utilized in exhibits of the drawing.

Prerequisite: a bountry polyline
Keyboard Command: hatch_boundary

Erase Boundary Hatch

This command erases the hatch drawn in the plan view.

Prerequisite: a boundry hatch
Keyboard Command: erase_boundary

Areas Of Interest

Function

Areas of Interest can be used to calculate volumes and material quantities within a specified area. The Area Of Interest perimeters are defined by user-selected closed polylines and each area is assigned a name. The Area Of Interest polylines can be assigned either as inclusion or exclusion perimeters for the area. You can have any number of exclusion perimeters within an inclusion but inclusion perimeters cannot be inside exclusions.

The Areas Of Interest (AOI) commands allow you to Tag/Untag Areas of Interest, Identify/Report Areas of Interest and Hatch/Label Areas of Interest.

Tag Area Of Interest

This command allows the user to select polylines and exclusion perimeters that define phases of a project. Carlson Takeoff will separate each area of interest in the volume and material reports.

Prerequisite: a desired polyline
Keyboard Command: tag_aoi

Area Of Interest by Interior Text

This command allows the user to select text from the screen to name AOIs and linework to determine the area.

Prerequisite: area linework and text
Keyboard Command: txt2aoi

Untag Area Of Interest

This command allows the user to remove previously tagged areas.

Prerequisite: an area of interest
Keyboard Command: untag_aoi

Identify Area Of Interest

This command allows users to identify AOI by either picking on a polyline(s) or by searching the entire drawing. The command will then report the AOI name, layer, type, starting point, and highlight the polyline in the plan view.

Prerequisite: an area of interest
Keyboard Command: id_aoi

Report Area Of Interest Areas

Use this command to report the Inclusion or Exclusion area (SF), the name, the layer, and the starting point.

Prerequisite: an area of interest
Keyboard Command: report_aoi

Hatch Area Of Interest Areas

This command allows the user to visually see AOIs in the plain view.

Prerequisite: an area of interest
Keyboard Command: hatch_aoi

Erase Area Of Interest Hatch

This command erases AOI hatching.

Prerequisite: hatched area of interest
Keyboard Command: erase_aoi_hatch

Label Area Of Interest Areas

This command labels the AOI name and area in the plain view.

Prerequisite: an area of interest
Keyboard Command: label_aoi

Erase Area Of Interest Labels

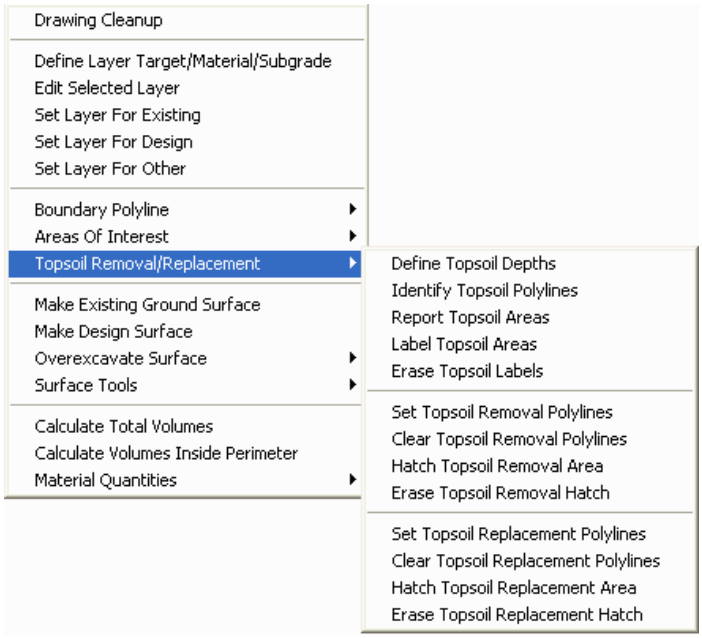
This command erases AOI labeling.

Prerequisite: hatched area of interest
Keyboard Command: erase_aoi_labels

Topsoil Removal and Replament Options

Function

The Topsoil Removal and Replacement options (shown here) allow the user to Define Topsoil removal and replacement depths, Set topsoil removal and replacement areas by selecting closed polylines, Clear the selected boundary polylines if needed, Hatch the topsoil removal and replacement areas and Erase the hatched areas.



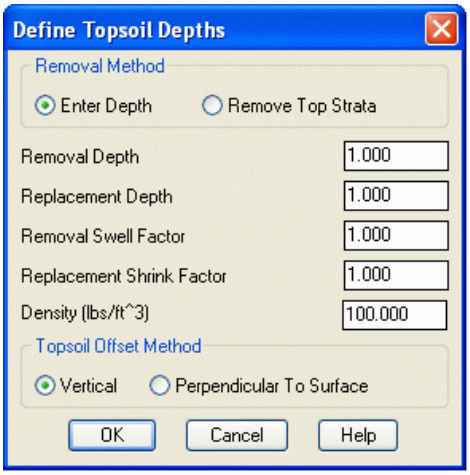
Define Topsoil Depths

This command requires user input to define the depth, or strata, of topsoil removal and replacement. Fill in the options available in the Define Topsoil Depths dialog (shown here). Carlson Takeoff will preform four functions with these values. First, the value set for the Removal Depth, or the Top Strata if selected, will be the "defined" removal amount from the Existing Ground Surface. Second, the calculated volume of topsoil removed will be included in the reporting options. Third, the value set for the Replacement Depth will be added "BELOW" the Finished Ground Surface model. Fourth, the amount of topsoil replaced will be included in the reporting options.

When topsoil depths are defined, the volume report routines will include the topsoil quantities. These topsoil quantities are in addition to the cut/fill for the existing to design surfaces for the site.

The Removal Swell Factor and Replacement Shrink Factor are multiplied by the topsoil removal and replacement quantities respectively in the volume report routines. The Density is used to report topsoil tons when the volume report option for tons is active.

The Topsoil Offset Method choose between offseting the topsoil depth vertically or perpendicular to the surface. The perpendicular method will result in more topsoil quantities since it represents applying the topsoil depth to the slope area of the surface whereas the vertical method represents applying the topsoil depth to the horizontal area.



Prerequisite: topsoil depths
Keyboard Command: define_topsoil

Identify Topsoil Polylines

This command allows users to identify topsoil polylines by either picking on a polyline(s) or by searching the entire drawing. The command will then report the layer name and starting point for both removal and replacement polylines. These polylines are also highlighted in the plain view.

Prerequisite: topsoil polylines
Keyboard Command: id_topsoil

Report Topsoil Areas

Use this command to report the Inclusion or Exclusion area (SF), the type, the depth, the layer, and the starting point.

Prerequisite: topsoil areas
Keyboard Command: report_topsoil

Label Area Of Interest Areas

This command labels the topsoil type and area in the plain view.

Prerequisite: topsoil area
Keyboard Command: label_topsoil

Erase Area Of Interest Labels

This command erases topsoil labeling.

Prerequisite: hatched area of interest
Keyboard Command: erase_topsoil_labels

Set Topsoil Removal Polylines

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil removal and any "CLOSED" interior polylines that will not have topsoil removed. The outer boundary usually is the same polyline that defined the "Boundary Polyline" selected earlier. The internal polylines usually are those that have been selected as the "Exclusion Polylines". The layer names for these boundarys is irrelevant. You will be prompted to use the Removal Depth defined in the Define Topsoil Depths command or to customize your depth.

Prerequisite: polylines for removal
Keyboard Command: tag_topsoil_remove

Clear Topsoil Removal Polylines

This command allows the user to remove and previously selected Topsoil Removal Polyline boundarys.

Prerequisite: topsoil polylines
Keyboard Command: untag_topsoil_remove

Hatch Topsoil Removal Area

Use this command to display a hatch pattern over the entire area designated for topsoil removal.

Prerequisite: topsoil areas
Keyboard Command: hatch_topsoil_remove

Erase Topsoil Removal Hatch

Use this command to remove the hatch pattern that defined the topsoil removal area.

Prerequisite: hatched topsoil
Keyboard Command: erase_topsoil_remove

Set Topsoil Replacement Polylines

Use this command to select the "CLOSED" polyline boundary defining the extents of topsoil replacement, and any "CLOSED" interior polylines that will not have topsoil replaced. The layer names for these boundarys is irrelevant. You will be prompted to use the Topsoil Replacement amount defined in the Define Topsoil Depths command or to customize your amount.

Prerequisite: polylines for replacement
Keyboard Command: tag_topsoil_replace

Clear Topsoil Replacement Polylines

This command allows the user to remove and previously selected Topsoil Replacement Polyline boundarys.

Prerequisite: topsoil polylines
Keyboard Command: untag_topsoil_replace

Hatch Topsoil Replacement Area

Use this command to display a hatch pattern over the entire area designated for topsoil replacement.

Prerequisite: topsoil areas
Keyboard Command: hatch_topsoil_replace

Erase Topsoil Replacement Hatch

Use this command to remove the hatch pattern that defined the topsoil replacement area.

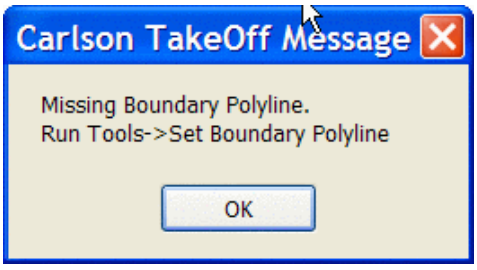
Prerequisite: hatched topsoil
Keyboard Command: erase_topsoil_replace

Make Existing Ground Surface

Function

This command makes the triangulation models for the existing ground surface. There are three surfaces that are created: initial original ground, original ground after applying subgrade zones, and original ground after subgrade zones and topsoil removal. These surface files are automatically named as "filename-og.tin", "filename-ze.tin" and "filename-ex.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangultion and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Existing surface, then the original ground after subgrades surface with be the same as the initial original ground surface. The topsoil removal depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil removal areas will lower the ground surface by the topsoil depth. If there aren't any topsoil removal areas, then the original ground after subgrade and topsoil surface will be the same as the original ground after subgrade surface.

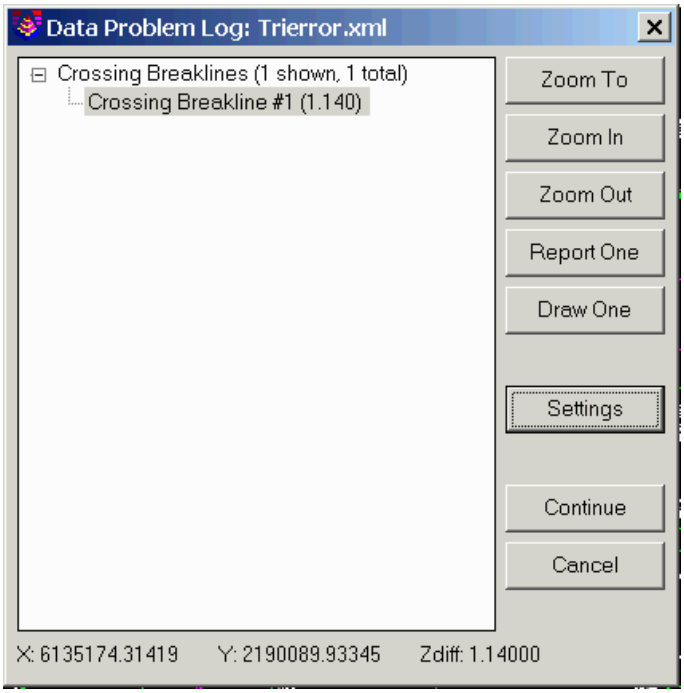
Before running this command, the layer names for the entities on the Existing layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site. If the boundary has not been set, the following error message will appear.



If this error message appears, run the "Set Boundary Polyline" command and pick the CLOSED polyline representing the boundary of the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Three types of conflicts are reported: Crossing Breaklines, Vertical Edges, and Breakline T-Intersections. Crossing Breaklines indicates that the intersection of two entities does not have a common elevation. Vertical Edges indicates that two entities or vertices of differing elevations have the same x-y location, thus forming a vertical plane. Breakline T-Intersections indicates that a 3d entity is abutting another entity, but the second entity doesn't have a vertex at the point of intersection. Each type of conflict is listed in its own category.

The Data Error Log shows the amount of elevation difference at each error. You can use the Data Error Log to review, report and draw markers at these error locations. Then you can exit the Data Error Log and fix the data errors with the commands in the Elevate menu or other drafting tools. After these errors are fixed, you can run Make Existing Ground Surface again.



Clicking to the "plus" sign beside a category will display the individual conflicts within that category. When a line item error is selected, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Zoom functionality allows the user to more closely inspect the specific problem area, and if needed a marker can be drawn or a report generated for an individual conflict or conflicts.

Zoom To pans the drawing to move the selected conflict to the center of the screen. The zoom functions are only active when a single line item is selected.

Zoom In zooms in on the highlighted area for closer inspection**.** Multiple picks on the zoom button will increase the magnification.

Zoom Out zooms out away from the highlighted area.

Report All/One toggles between One and All depending whether a single line item conflict or a category is selected from the error log. An error report is generated listing the x-y position and the elevation difference of the entities in conflict.

Draw All/One toggles between One and All depending whether a single conflict or a category is selected from the list. This option draws an "X" symbol at each selected conflict. The layer and size of the symbol is controlled in the fields below.

Continue closes the Error Log and proceeds with the contouring operation.

Settings has controls for the tolerances for error reporting and for the Layer Name and Symbol Size to use with the Draw function.

Prerequisite: a boundary polyline and elevated entities on the Existing layer target
Keyboard Command: mk_exist_tin

Make Design Surface

Function

This command makes the triangulation models for the design surface. There are three surfaces that are created: initial design, design after applying subgrade zones, and design after subgrade zones and topsoil replacement. These surface files are automatically named as "filename-bs.tin", "filename-zn.tin" and "filename-fn.tin" respectively. The "filename" is set to the name of the current drawing (dwg) file. Also, the file extension will be .tin for the binary format triangultion and .flt for the ASCII format triangulation. This file format is set in Configure->Takeoff. The subgrade zones are defined in the Define Layer Target/Material/Subgrade command. If there aren't any subgrade zones for the Design surface, then the design after subgrades surface with be the same as the initial design surface. The topsoil replacement depths and areas are set with the commands in the Topsoil Removal/Replacement sub-menu. The topsoil replacement areas will lower the design surface by the topsoil depth to leave room for the topsoil replacement. If there aren't any topsoil replacement areas, then the design after subgrade and topsoil surface will be the same as the design after subgrade surface.

Before running this command, the layer names for the entities on the Design layer target must be set in the Define Layer Target/Material/Subgrade command. Also these entities must be at their proper elevations. The entity elevations can be reviewed using commands from the Inquiry menu and the elevations can be assigned if needed using command from the Elevate menu. Another prerequisite is that the Boundary Polyline must be set for the site.

When the program finds errors in the existing entities, a Data Error Log dialog reports these errors. Refer to the Make Existing Surface command for more information on the Data Error Log dialog.

Prerequisite: a boundary polyline and elevated entities on the Existing layer target

Keyboard Command: mk_final_tin

Adjust Design Surface

Function

The commands in this sub-menu adjust the design surface.

Vertical Offset

Use this command to vertically move the design surface model up or down. Carlson Takeoff offers vertical adjustments to the entire site, or to designated areas of the site represented by selecting CLOSED polyline boundaries. To vertically move the entire site, or select areas of the site, "UP", enter a positive value in feet and/or tenths of feet. A negative value will move the site down. For example, a value of 1.1 would move the site up 1.1 feet.

Merge With Existing

This command merges the existing ground surface model onto a portion of the design surface. This routine can be used when a section of the site is completed and you want to set part of the model as done. The area to merge is defined by selecting inclusion polylines which must by closed polylines. There is also the option to select exclusion polylines for areas not to merge.

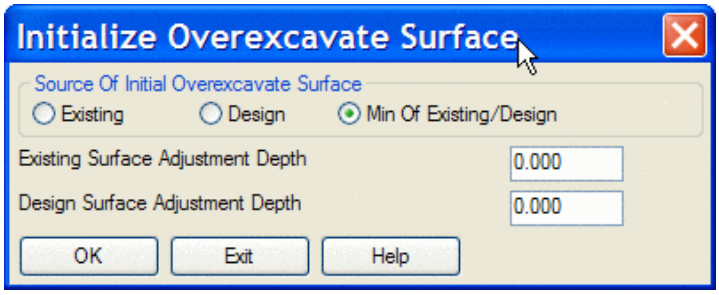
Prerequisite: design surface
Keyboard Command: adjust_final

Overexcavate Surface

Make Overexcavation Surface From Exsting/Design Surface

Function

The Initialize Overexcavation Surfaced dialog box shown here allows the user to select which surface model to overexcavate and to enter in the depth value for the desired adjustment. Use the Min Existing/Design option to set the overexcavate as the minimum of the existing and design surfaces. If a single surface is selected the value entered will be applied to that surface only. The resulting surface of Make Overexcavate Surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.

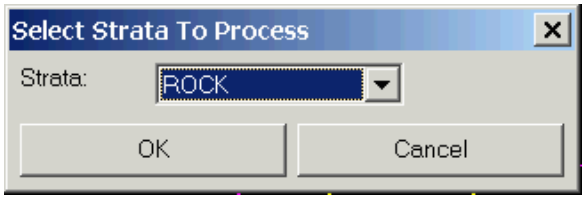


Prerequisite: Existing and/or Design surfaces
Keyboard Command: set_overx

Make Overexcavation Surface From Strata Surface

Function

This command sets the Overexcavate surface to a selected strata surface. Before running this command, the strata surface must be created with the Make Strata Surfaces command in the Drillhole menu. The resulting overexcavate surface is stored in a triangulation file that is named with "-ox" appended to the current drawing name.



Prerequisite: Strata surfaces
Keyboard Command: overx_by_strata

Make Overexcavate Surface From Screen Entities

Function

This command makes the overexcavate surface from entities on the layers defined as Overexcavate in the Define Layer Target/Material/Subgrade command. The resulting surface of Make Overexcavate Surface is stored in a

triangulation file that is named with "-ox" appended to the current drawing name.

Prerequisite: overexcavate entities
Keyboard Command: mk_overx_tin

Adjust Overexcavate Surface

Function

This command adjusts the overexcavate surface vertically within the selected perimeter polylines. This command allows the site to be overexcavated at a variety of depths in specified areas represented with CLOSED polyline boundaries. Select the desired areas to be adjusted when prompted at the command line.

Prerequisite: an overexcavate surface
Keyboard Command: adjust_overx

View Overexcavate Surface

Function

Use this command to view the current overexcavate surface. The Takeoff 3D Viewer will display the 3D faces of the adjusted surface. Shade the 3D model and adjust its perspective to view a rendered display. The surface that is displayed will depend on the latest surface created using the make and adjust routines.

Prerequisite: an overexcavate surface
Keyboard Command: cube_overx

Draw Overexcavate Surface 3D Faces

Function

Use this command to draw the 3D faces of the overexcavated surface model on the screen. The 3D faces will be drawn in the TK_OVERX_SURFACE layer and will depend on the latest surface created using the make and adjust routines.

Prerequisite: An overexcavate surface
Keyboard Command: draw_overx

Erase Overexcavate Surface 3D Faces

Function

Use this command to remove the previously drawn 3D Faces from the screen.

Prerequisite: 3D Faces
Keyboard Command: erase_overx

Draw Overexcavate Cut Color Map

Function

Use this command to display a cut color map on the screen that shows the areas of overexcavate cut. The colors will graduate from white to red based on zero cut depth to maximum cut depth. This command also offers the user to place a legend of the cut depths on the screen. Pick the desired location and type the desired scale of the legend when prompted at the command line.

Prerequisite: An overexcavate surface
Keyboard Command: overx_cfmap

Erase Overexcavate Cut Color Map

Function

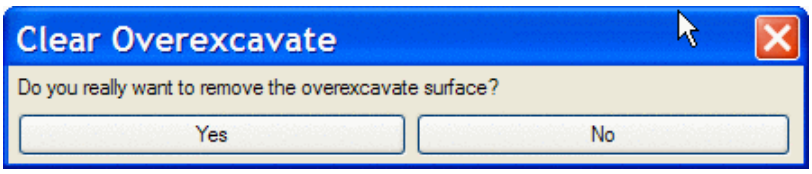
Use this command to remove the previously drawn Cut Color Map and Legend from the screen.

Prerequisite: An overexcavate cut color map
Keyboard Command: overx_cfmap2

Clear Overexcavate Surface

Function

Use this command to remove the overexcavate surface. When the overexcavate surface is removed, the rest of the Takeoff commands will not calculate overexcavate volumes. You will be prompted to confirm before the remove is done.



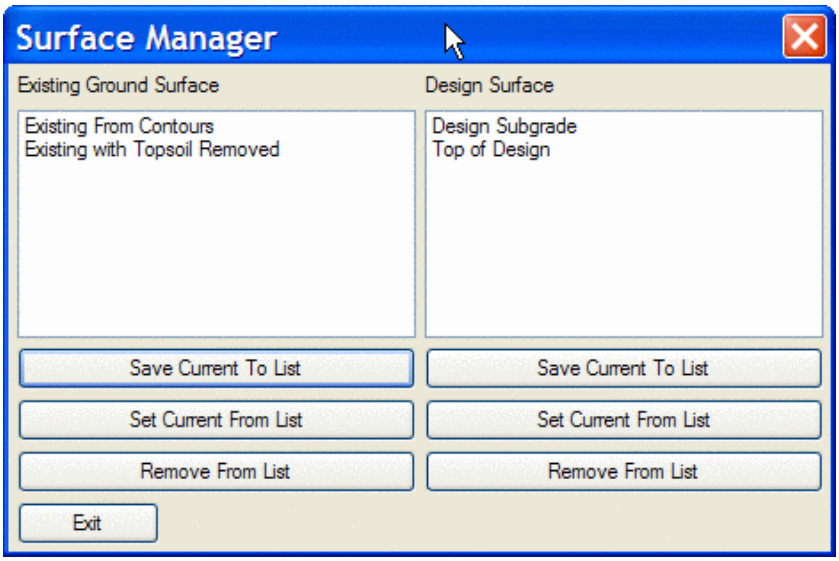
Prerequisite: An overexcavate surface

Keyboard Command: clear_overx

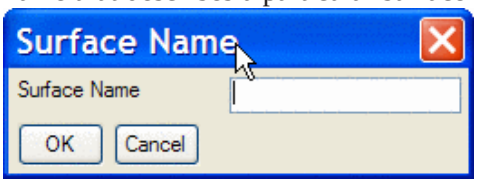
Surface Manager

Function

This command allows the user to name and manage multiple surface models. The Surface Manager dialog shown here has options to name and save the current "existing and design" surface models. The "current" surface is dictated by the layers that populate a target and the Make Surface command. If layers are removed from a target, and others assigned, multiple surfaces can be created and stored. Highlight a named surface and select the Set Current From List option to make that model active. Use the Remove From List option to remove a named surface model from the list.



Selecting the Save Current To List options brings up the Surface Name dialog box shown here. Type the desired name that describes a particular surface model and select OK.



Prerequisite: none
Keyboard Command: surf_mgr

Make User-Defined Surface

Function

This command makes a surface from the entities on the layers defined as user-defined targets in the Define Layer Target/Material/Subgrade command. The purpose of user-defined surfaces is for modeling surfaces besides existing ground and design. The drawing needs to contain entities that represent the elevations of the user-defined surface. For example, the user-defined surface could be for alluvial soil and the drawing has contour polylines for this surface.

There is a dialog to select which surface to make. The surface is stored in a triangulation file that is named after the current drawing name with the user-defined surface name appended.

This user-defined surface can be applied to Takeoff routines by running the Set Active Surfaces command.



Prerequisite: Define Layer Target/Material/Subgrade command
Keyboard Command: mk_user_tin

Set Active Surfaces

Function

This command assigns which surfaces to use for initial and final. These surfaces are used by all the Takeoff routine that compare surfaces including:
- Calculate Total Volumes
- Calculate Volumes Inside Perimeter
- 3D Drive Simulation
- Cut/Fill Contours/Labels/Color Map
- Surface Inspector
- Quick Profile
- etc.

The surface created by the Make Existing Ground Surface command is called "Existing" and is the default for the Initial Surface. The surface created by the Make Design Surface command is called "Design" and is the default for the Final Surface.

The purpose of this routine to for selecting user-defined surfaces to use in place of the existing ground or the design surface. For example, there could be a user-defined surface for alluvial soil that is set as the initial surface

while design is set to the final surface. Then the calculate volume routines will report the quantities between alluvial soil and design. Also the Display->Cut/Fill Color Map routine will make the map for the difference between the alluvial soil and design surfaces.

These user-defined surfaces can be created using the Add Target function in the Define Layer Target/Material/Subgrade command combined with the Make User-Defined Surface command.

Prerequisite: a suface model
Keyboard Command: set_active_tins

Design Surface Vertical Offset

This command can be used to lower or raise the design surface within a defined perimeter or by the entire surface.

Prerequisite: a design surface
Keyboard Command: adjust_final

Existing Surface Vertical Offset

This command can be used to lower or raise the existing surface within a defined perimeter or by the entire surface.

Prerequisite: an existing surface
Keyboard Command: adjust_exist

Merge Existing With Design

This command allows you to merge the existing surface with design surface within perimeter polylines. The resulting merged surface can be saved to update either the Existing or Design surfaces. The program prompts for inclusion and exclusion perimeter polylines. These polylines must be closed. The merge will be applied inside the inclusion perimeters and not inside the exclusion perimeters. The exclusion perimeters are optional.

For example, if a portion of the site is completed, you can update the existing surface to match the design for the completed area. First, draw a closed polyline around the completed area. Then run Merge Existing With Design and choose the merge results target as Existing. Then select the perimeter polyline.



Prerequisite: existing and design surfaces and an inclusion perimeter polyline
Keyboard Command: merge_final

Triangulate & Contour

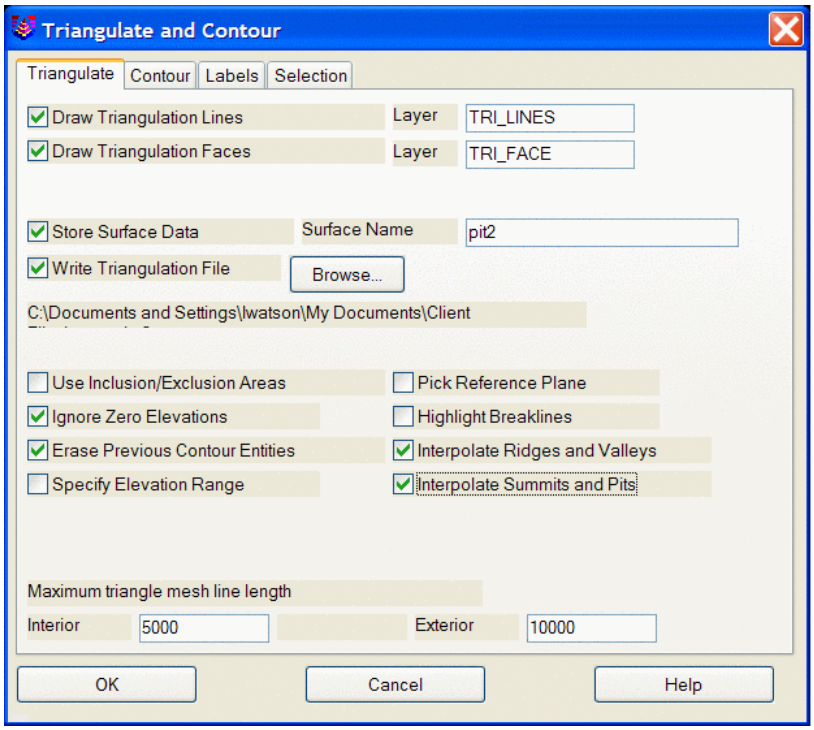
Function

This command provides all of the functionality related to contouring and creating tin surface models in one routine. Given data entities that represent the surface, this command creates a final contour map with labeled, smoothed, and highlighted contours and/or a surface model that can be saved to a file (to be used in other areas of the program) or drawn on the screen as triangles or faces. Eligible data entities include points, inserts, lines, 2d polylines, 3d polylines, elevation text, 3d faces, and points from ASCII or coordinate (.CRD) files.

Triangulate & Contour has many options which are defined in the exhibits shown in the following pages. With this command, you can do any combination of drawing the triangulation network lines, drawing the contours, drawing triangulation network 3D Faces or lines, writing a triangulation file and storing a surface file.

In order to force *Triangulate & Contour* to correctly interpolate elevations between two points that define a grade break in the surface (such as points on a ridge, wall, or road), a breakline must exist between the points. A breakline line can be specified as a 3D polyline or line. In fact, all 3d polyines and lines with elevatioin are be treated as breaklines.

Triangulate Tab



Draw Triangulation Lines

When this option is turned on, the program will draw the triangulation as 3D lines. Specify the layer for these lines in the box to the right.

Draw Triangulation Faces

When this option is turned on, the program will draw each triangle in the triangulation network as a 3D Face. These 3D Faces can then be used in AutoCAD's modeling routines such as *HIDE* and *SHADE* or in routines such as *3D Viewer Window, 3D Surface FlyOver and Slope Zone Analysis*. Specify the layer for these 3DFaces in the box to the right.

Store Surface Data

This option names and creates a surface or surfaces that are associated with the drawing. The creation of a surface is necessary in order for the Surface Tools to function. A Triangulation file must also be specified before using the Store Surface option.

Write Triangulation File

This option stores the triangulation surface model as an .flt or a .tin file. The .flt file format is a text file depicting the edges in the triangulation network. The .tin file is a new binary file format depicting the triangualtion network. The .tin file is much faster and more efficient than the previous .flt file format. The triangulation file/s can be used by several commands such as *Volumes By Triangulation, Spot Elevations, and Profile from FLT File*. Either type in the file name to create or press the Browse button to select a file name.

Use Inclusion/Exclusion Areas

When this box is activated, the program will later prompt you for inclusion and exclusion polylines which are used to trim the contours. The inclusion and exclusion polylines must be closed polylines and must be drawn before starting *Triangulate & Contour*. Only the parts of the contour lines that are within the inclusion polylines will be drawn. For example, an inclusion could be the perimeter of the site. The parts of contour lines that are inside the exclusion polylines are not drawn. Exclusion polylines can be used for areas where you don't want contours such as within buildings.

Ignore Zero Elevations

When activated, this setting will filter out all data points at an elevation of zero from the data set.

Erase Previous Contour Entities

When activated, this setting will erase previously drawn contour entities.

Specify Elevation Range

The program will automatically contour from the lowest elevation in the data set up to the highest at the increment specified in Contour Interval. If you would like to manually set the range over which to contour, select this option.

Pick Reference Plane

The triangulation network is based on the x,y position of the points. This option allows you to contour an overhang or cliff by changing the reference plane to a side view. The reference plane can be specified by first using the *Viewpoint 3D* command and then using the View option, or you can specify three data points on the cliff (two along the bottom and one at the top).

Highlight Breaklines

This option highlights breaklines in the triangulation network by drawing the triangulation lines along breaklines in yellow.

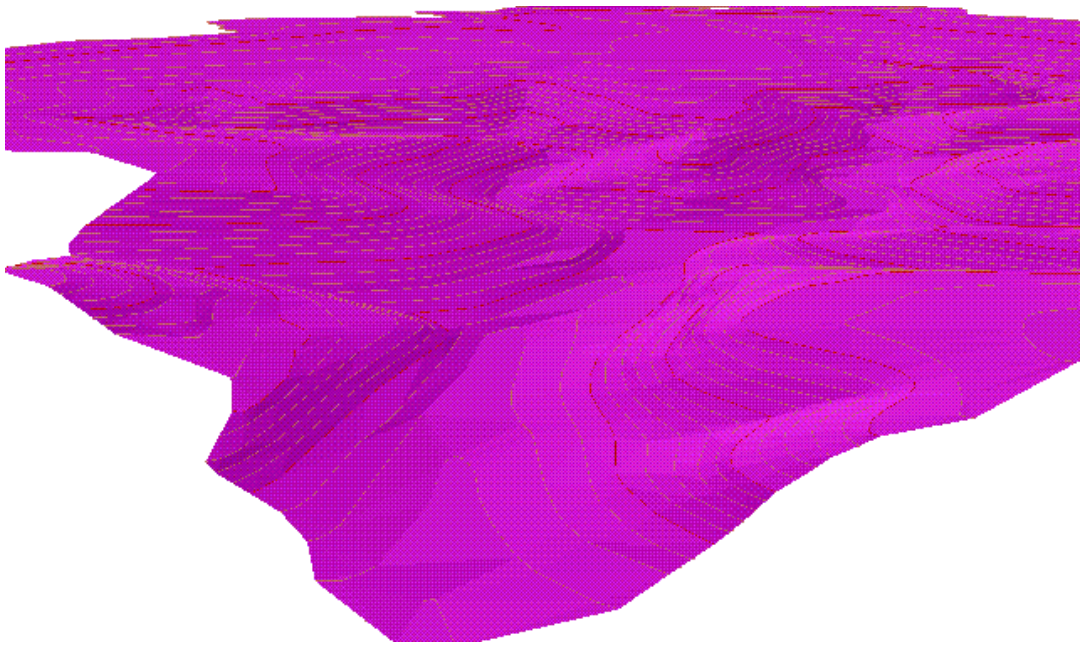
Interpolate Ridges and Valleys

This option creates additional triagulation in a ridge or valley situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours, since it replaces the need to manually draw 3d polylines along ridges and valleys.

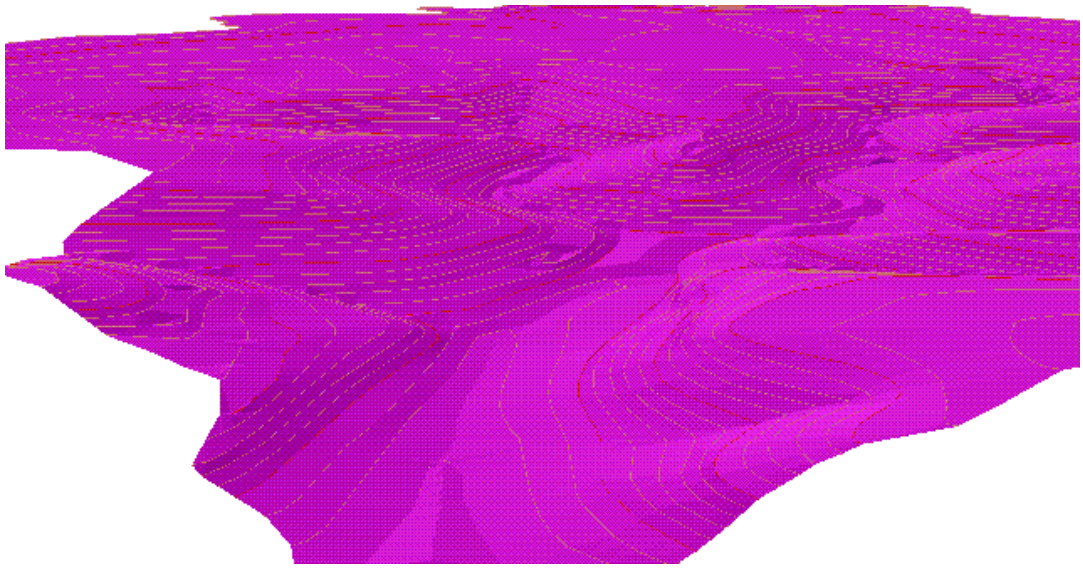
Interpolate Summits and Pits

This option creates additional triagulation in a summit or pit situation to more accurately define the feature during surface modeling operations. This option would commonly be used when creating a surface model from existing contours.

Before: Surface made from an existing contour map. Note the flat spots in the bottom of the valley when Interpolate Ridges and Valleys is disabled.



After: The same surface with Interpolate Ridges and Valleys enabled.

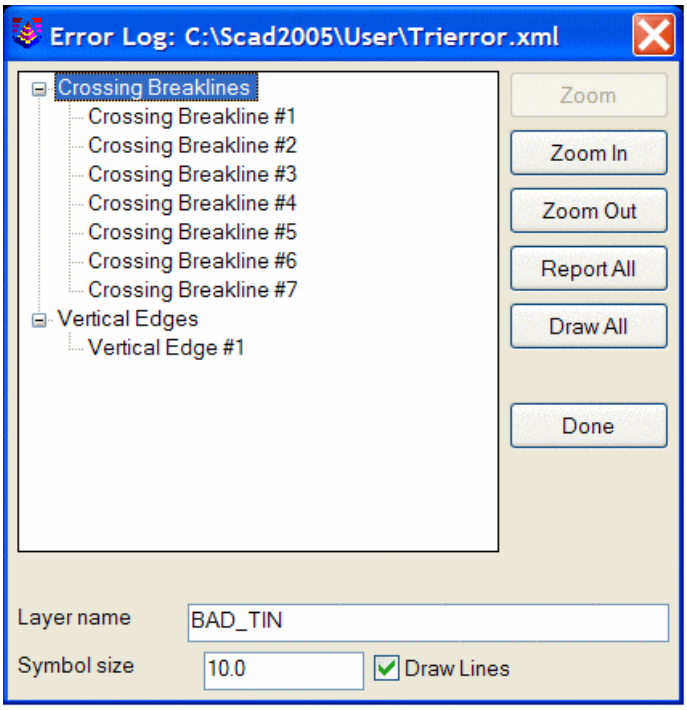


Max Triangle Mesh Line Length

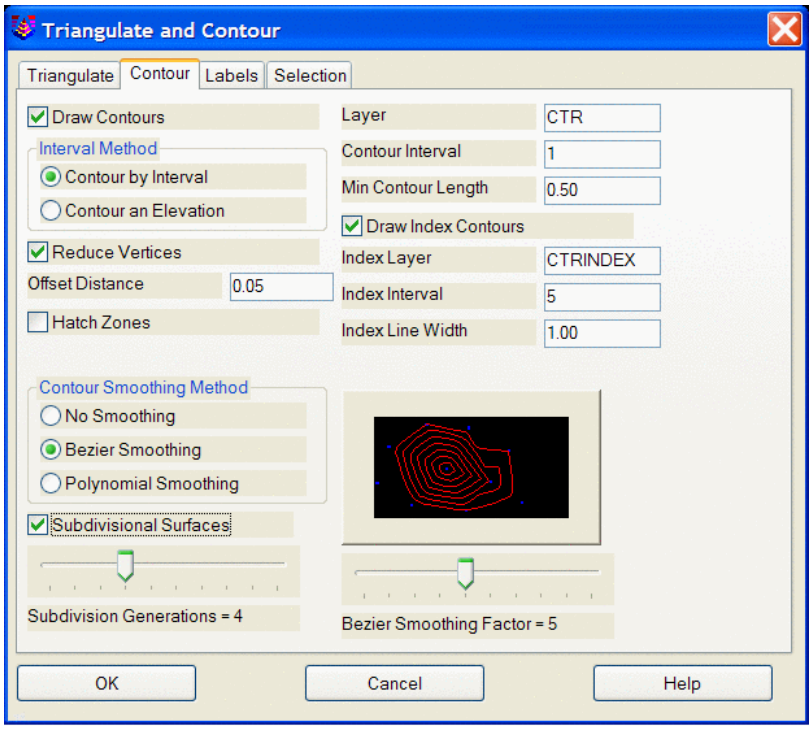
This value limits the length of the triangulation network lines. Any triangulation line that exceeds this limit will not be drawn or included in contouring. This allows you to avoid abnormally long triangulation lines where you have relatively too few data points and on the outskirts of your data points. The **Exterior** value applies to triangulation lines around the perimeter of the triangulation area and the **Interior** value applies all the other triangulation lines. Generally you would have the exterior value larger than the interior.

Error Log

The following dialog box appears when the Triangulate & Contour routine finds a conflict between breaklines or other surface entities. The type of conflict is identified, and when an item is chosen, a highlighted arrow is temporarily placed in the drawing to indicate the exact location of the specific conflict. Crossing Breaklines indicates that the intersection of two entites has two differing elevations. Verticle Edges indicates that two entities or vertexes of differing elevations have the same xy location, thus forming a vertical plane.



Contour Tab



Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or **Contour an Elevation**
Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Verticies option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

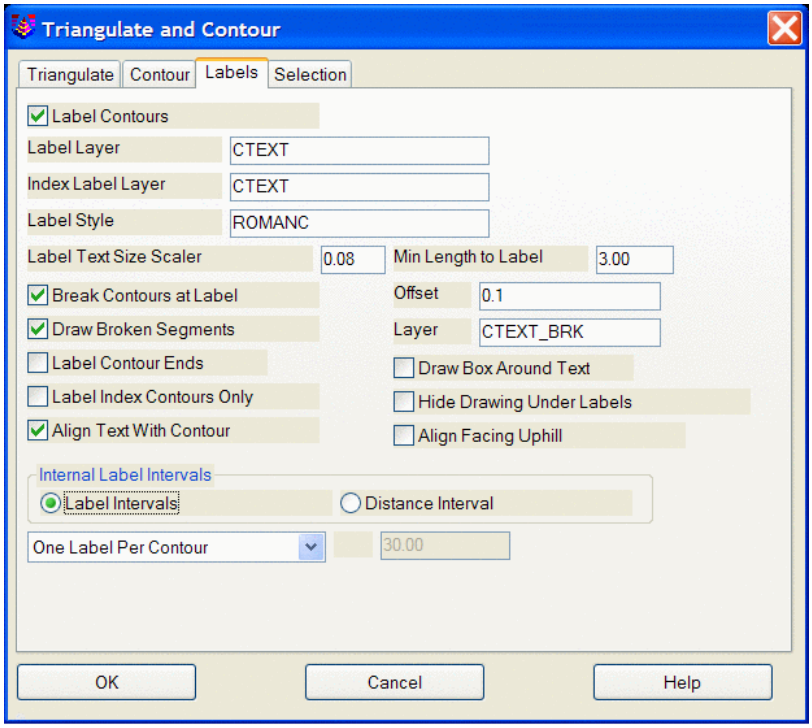
Bezier Smoothing Factor

The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation

This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Labels Tab



Label Contours

When activated, contours will be labeled based on the settings below.

Label Layer

Specifies layer name for intermediate contour labels.

Index Label Layer

Specifies layer name for index contour labels.

Label Style

Specifies the text style that will be used for the contour label text.

Label Text Size Scaler

Specifies the size of the contour labels based on a multiplier of the horizontal scale.

Min Length to Label

Contours whose length is less than this value will not be labeled.

Break Contours at Label

When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right acitvates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Broken Segments

When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends

When checked, contour ends will be labeled.

Draw Box Around Text

When checked, a rectangle will be drawn around contour elevation labels.

Label Index Contours Only

When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels

This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity of the contour label.

Align Text with Contour

When checked, contour elevation labels will be rotated to align with thier respective contour lines. This option also acitvates the Align Facing Uphill feature explained below.

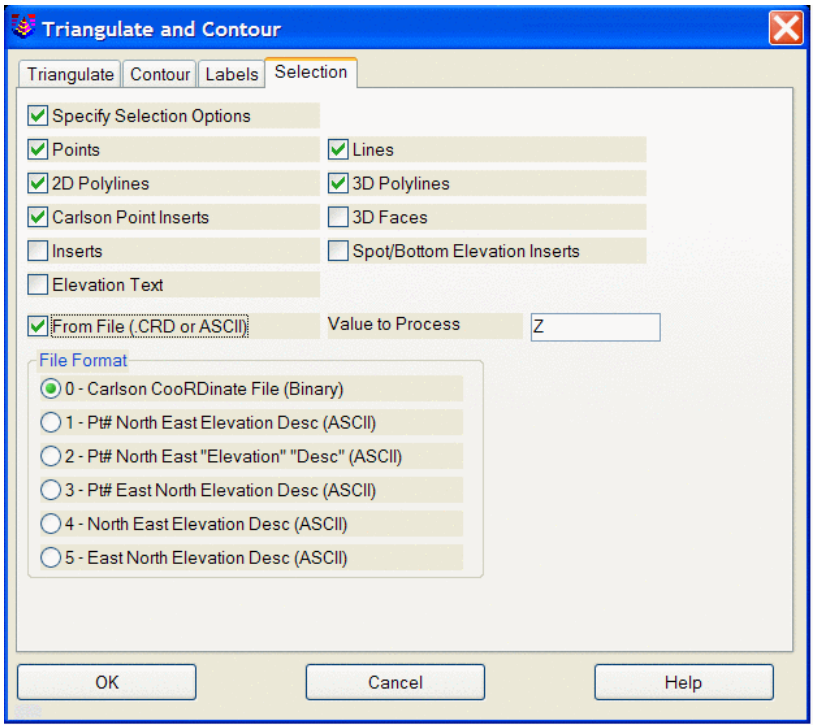
Align Facing Uphill

When checked, contour elevation labels will still be rotated to align with thier respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

Internal Label Intervals

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.

Selection Tab



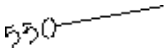
Specify Selection Options

When checked, this allows you to control what type of entities *Triangulate & Contour* uses.

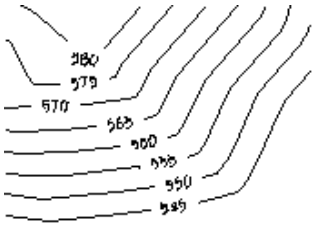
Points, 3D Polylines, 2D Polylines, Lines, Inserts are standard AutoCAD entities types.

Spot/Bottom Elevation Inserts include text entities that start with 'X'.

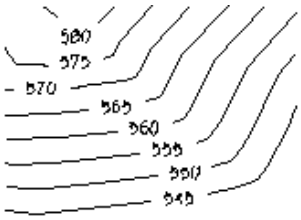
From File allows you to triangulate from the points in a coordinate (.CRD) or ASCII file.



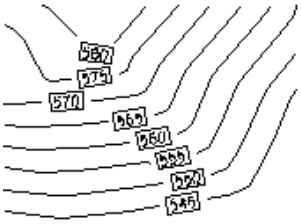
Label Contour Ends



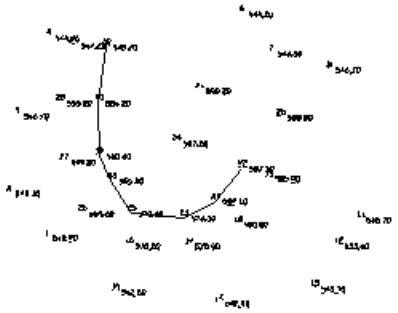
Align Text With Contour ON

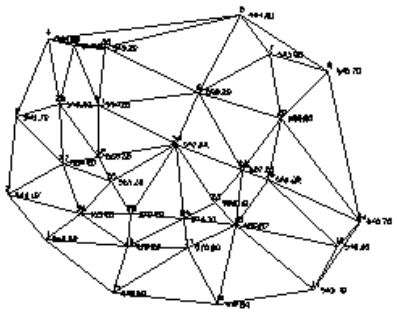


Align Text With Contour OFF

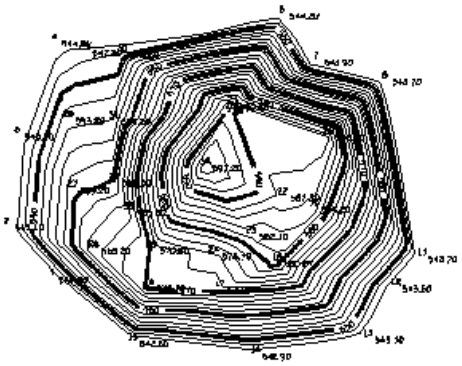


Draw Box Around Text





Triangulation network without contouring
Original data points with one 3D polyline



Contours without triangulation network. The contours are smoothed, reduced, drawn at an interval of 2, and highlighted at an interval of 10 with labeling on the index contours.

Pull-Down Menu Location: Tools-> Surface Tools

Prerequisite: Data points of the surface

Keyboard Command: tri

Triangulation File Utilities

Function

This command allows you to modify TIN surfaces in a variety of different ways, then allows for 3d viewing and shading of the modified surface and finally for saving the file with a choice of output formats. The focus of the routine is to elevate or lower the TIN or selected areas within the TIN, merge TINs with other surfaces, or use data from other TIN files to apply to the current TIN. Operations can be performed on the entire TIN or just on user selected Inclusion and/or Exclusion areas. The routine will automatically rework the TIN network for conformation to a selected boundary, say a building outline. In the case of said building, a value of 10 could be subtracted from the building outline. This will drop all of the triangulation within the outline by 10', thus creating a model of the

excavated area for the building. The modified TIN can then be saved to a new file, which could be used to compute an excavation volume with Volumes by Triangulation. This routine does not allow for manual reconfiguration of the TIN network. This is performed under Surface Tools, also in the Contour pulldown menu. This routine also includes conversions to and from TIN files, DXF files and 3D Face entities.

Begin with the dialog shown here. First select a TIN model. You may choose between an .flt or .tin file, a DXF file (that includes 3DFACE entities), or 3DFACE entities in the current drawing. Specify the subject area by choosing inclusion or exclusion perimeters, then press the next button.

Load TIN File: Allows you to specify a triangulation (.flt or .tin) file to load.

Load DXF File: Allows you to specify a DXF file to load. Only loads 3DFACE entities from the selected DXF file.

Select 3D Faces: Allows you to select 3DFACE entities from the current drawing. This also includes rectangular 3d faces from a plotted grid.

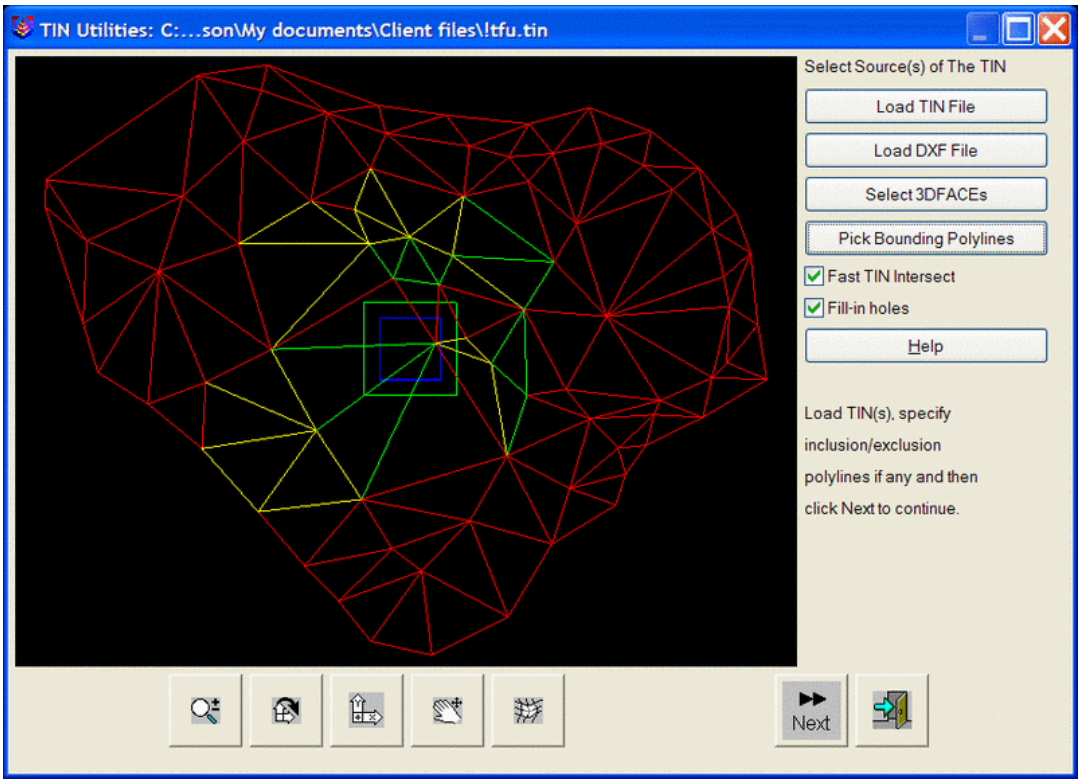
Pick Bounding Polylines: Allows you to select any inclusion/exclusion perimeter(s). When this button is selected, the user is taken back to the drawing and prompted to select the perimeters. Press Enter when the selections are finished to return back to the dialog.

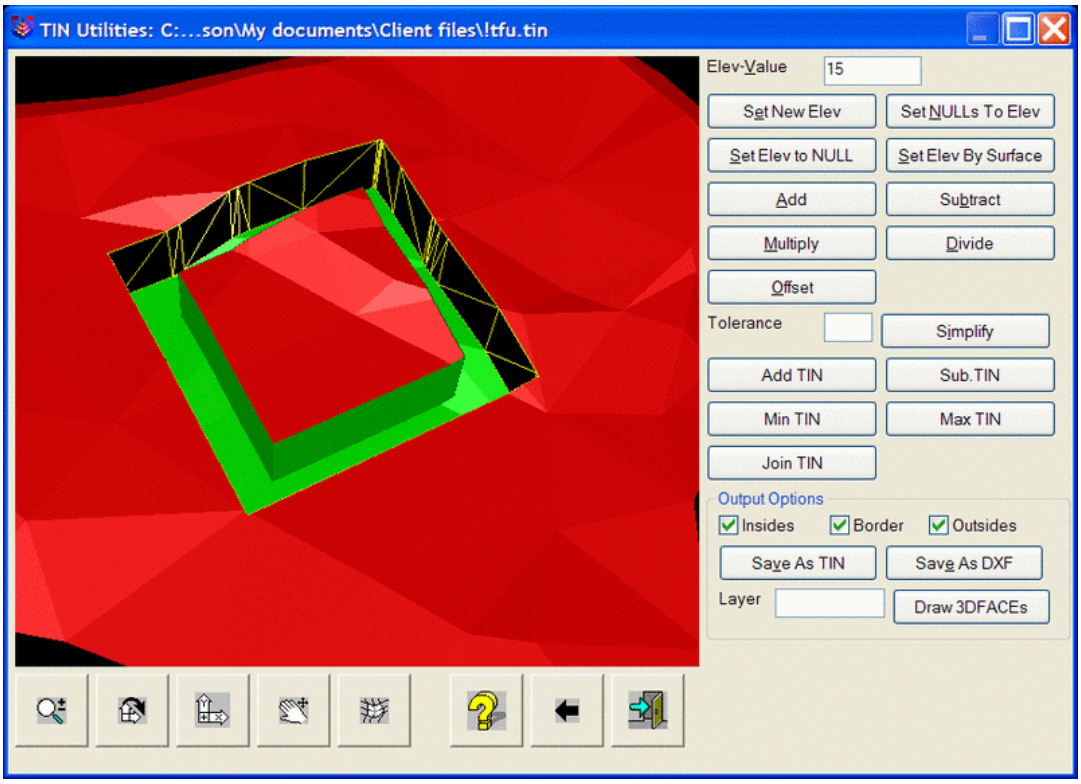
Fast TIN Intersect: When checked, this command will not try and intersect 3DFACE entities.

Fill-in-holes: When checked, any missing triangulation or gap in the surface will be automatically filled in with additional triangles. This option has to set before loading the TIN file to take effect.

Next: Press this button to proceed to the next dialog after all selections have been made .

The next dialog allows you to perform mathematical operation(s) on the loaded TIN. Each operation is described below. Keep in mind that generally these operations are to be performed on an area inside your inclusion perimeter (but excluding anything inside your exclusion perimeters). If you





do not specify any perimeters, the desired operation/s will be performed on the entire TIN.

Elev-Value: Specify either an elevation or value depending on the operation. The value specified will be used for subsequent operations.

Set New Elevation: Sets all TIN faces in the subject area to the elevation specified in the Elev-Value field.

Set NULL's to Elevation: Sets all NULL values in the subject area to the elevation specified in the Elev-Value field.

Set Elevation's to NULL: Sets all of the elevation values in the subject area to NULL.

Set Elevation by Surface: Sets all TIN faces within the subject area to the elevations from a second surface file within the same area. You will be prompted to select a second TIN file or grid file. Only areas common to both surfaces will be applied to the subject TIN.

Add: Adds the value specified in the Elev-Value field to the subject area of the TIN.

Subtract: Subtracts the value specified in the Elev-Value field to the subject area of the TIN.

Multiply: Multiplies by the value specified in the Elev-Value field to the subject area of the TIN.

Divide: Divides by the value specified in the Elev-Value field to the subject area of the TIN.

Offset: Performs a perpendicular offset of the TIN surface by the specified amount.

Tolerance: This setting is used by the Simplify command described below. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

Simplify: Causes edges within the Tin mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Add TIN: Raises the subject area of the current TIN by the elevation value from a second user selected TIN file. This function is most applicable to applying a strata thickness TIN.

Subtract TIN: Lowers the subject area of the current TIN by the elevation value from a second user selected TIN file.

Min TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the lower value of the two TINs to the subject area.

Max TIN: This does a comparison between the current TIN and a second user selected TIN file, and applies the higher value of the two TINs to the subject area.

Join TIN: Merges the existing subject TIN into a second user selected TIN file. The subject TIN file should be the smaller of the two surfaces since the subject file will be joined or merged into the second file.

Insides: If this is the only option checked, only changes made within the inclusion perimeter will be saved. TIN entities outside of the perimeter will not be saved.

Border:

Outsides: If this is the only option checked, TIN entities inside of the inclusion perimeter will not be saved. Everything outside of the perimeter will be saved.

SaveAs TIN: Saves the current TIN as an .flt or .tin file.

SaveAs DXF: Saves the current TIN as a DXF file. This format can be used by many other CAD programs.

Draw As 3DFaces: Draws the current TIN as 3D Faces in the current viewport. The Layer window is used to specify the layer that the faces will be created in.



This icon converts the right mouse button to a zoom function. Hold the button down and move the mouse up or down to zoom in and out.



This icon converts the right mouse button to a rotate function. Hold the button down to rotate the view in any X, Y or Z direction. When the XY appears in the window, the rotation will occur relative to the XY axis. When the mouse is moved toward the outer perimeter of the window, the XY will change to a Z. Holding the button down while the Z is visible will rotate the drawing on the Z axis.



This icon converts the right mouse button to a pan function. Hold down on the button while moving the mouse to pan. Holding down the mouse wheel will also serve as a pan function in any of the above modes.



This icon toggles shading on and off.



This icon restores the graphics to plan view.



This icon reverses the effects of all operations performed on the TIN and reverts it back to its original status.



This icon exits the routine. If the TIN has been modified, you will be prompted to save.

Pull-Down Menu Location: Tools-> Surface Tools

Prerequisite: 3D Faces, a TIN file or a DXF file.

Keyboard Command: TINUTIL

Volumes By Triangulation

Function

Volumes By Triangulation is an alternative volume method that compares two triangulation networks. This method is different from the grid based volume routines (*Stockpile Volumes, etc.*) and the cross section volume routine (*Calculate Section Volume*). Volumes by Triangulation calculates faster in most cases than the other methods, and it is the most accurate because it uses true TIN to TIN primodal volumes. This added accuracy in general is very small. The grid resolution is usually sufficient to model the surface for the grid based volumes. The Volume By Triangulation accuracy applies well when there is a feature like a 5 foot wide ditch. Then the grid resolution would need to be less than 5 foot to model the ditch which might be difficult on a large site.

The disadvantage to this routine is that it lacks the output options that help the analysis of the volume such as Difference Contours. Also Volumes by Triangulation does no extrapolation and stops calculating volume at the perimeter of the smaller of the two triangulation networks. Volumes By Triangulation is better when used with point data instead of contour data because contour data requires triangulating all the contour polylines as breaklines which creates a large triangulation network and is slower.

The triangulation networks to compare are defined in .flt files that are created by *Triangulate & Contour* with the Write Triangulation File option. Before using this command, run *Triangulate & Contour* twice to create an triangulation (.TIN) file for each surface. The volume calculation is limited by either the extent of the triangulation networks or by an inclusion/exclusion perimeter(s). These perimeters must be closed polylines.

Output data includes area, tons by density, average thickness, shrink and swell, ratio, and total volume.

Prompts

Select EXISTING surface Tmesh File

Choose a .tin file

Select FINAL surface Tmesh File

Choose another .tin file

Pick inclusion perimeter polyline (ENTER for none): *pick a closed polyline perimeter*

Calculating ...

Write report to file (Yes/<No>)? *Press Enter*

Write report to printer (Yes/<No>)? *Press Enter*

Comparing Triangulation files: C:\SCADXML\DATA\TRI1.FLT

and C:\SCADXML\DATA\TRI2.FLT

Cut volume: 66891.35 C.F., 2477.46 C.Y.

Fill volume: 43458.01 C.F., 1609.56 C.Y.

Pull-Down Menu Location: Tools-> Surface Tools

Prerequisite: Two .tin files.

Keyboard Command: trivol

Draw 3DPoly Perimeter

Function

This command draws a 3D polyline in the PERIMETER layer. This polyline is required by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines. In these routines, this polyline is used as the inclusion perimeter for volumes. You may want to set your AutoCAD Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline.

Prompts

Select the 3Dpolyline options you want and press enter.

Command: 3dperim
[Continue/Extend/Follow/Options/<Pick point or point numbers>]: (Pick)
Interpolate/screen Pick/<Elevation> <0.00>: 818

Z: 818.00
[Arc/Close/Direction/Follow/Undo/<Pick point or point numbers>]: (Pick)
Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 814

Z: 814.00, Hz dist: 60.64, Slope dist: 60.77, Slope: -6.6% Ratio: -15.2:1
[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Pick)
Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 815

Z: 815.00, Hz dist: 26.32, Slope dist: 26.33, Slope: 3.8% Ratio: 26.3:1
[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: c for close

Pull-Down Menu Location: Tools-> Surface Tools

Keyboard Command: 3dperim

Prerequisite: None

Draw 3DPoly Base Breakline

Function

This command draws a 3D polyline in the BASE_BREAKLINE layer. This polyline is used by the *Calculate Stockpile Volume* and *Calculate Pond/Pit Volume* routines to model the base surface. You may want to set your AutoCAD Object Snap prior to running this routine so that you obtain the elevations of existing points while creating the 3D polyline.

Prompts

Select the 3Dpolyline options you want and press enter.

command: 3DBASE
[Continue/Extend/Follow/Options/<Pick point or point numbers>]: (Pick)
Interpolate/screen Pick/<Elevation> <0.00>: 818

Z: 818.00
[Arc/Close/Direction/Follow/Undo/<Pick point or point numbers>]: (Pick)
Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 814

Z: 814.00, Hz dist: 60.64, Slope dist: 60.77, Slope: -6.6% Ratio: -15.2:1
[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Pick)
Slope/Ratio/Interpolate/Degree/screen Pick/<Elevation> <0.00>: 815

Z: 815.00, Hz dist: 26.32, Slope dist: 26.33, Slope: 3.8% Ratio: 26.3:1
[Arc/Close/Direction/Extend/Follow/Undo/<Pick point or point numbers>]: (Enter)

Pull-Down Menu Location: Tools->Surface Tools

Prerequisite: None

Keyboard Command: 3dbase

Calculate Stockpile Volume

Function

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is above the perimeter elevation, such as in the case of a stockpile of material. The complimentary command, *Calculate Pond/Pit Volume*, is for the opposite situation, in which the entire volume to be calculated is below the elevation of the perimeter.

The program internally computes base and final grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the Draw 3D Polyline Perimeter command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the base surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The final surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, but not including the BASE_BREAKLINE polylines.

You have the option of setting the resolution of the grids.

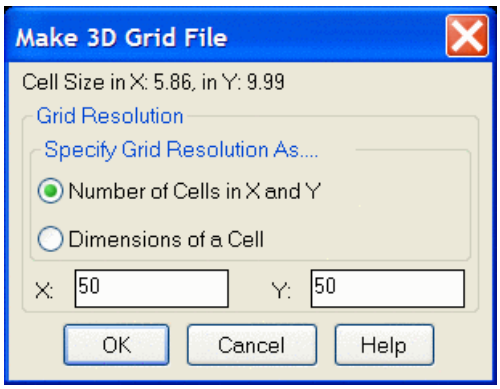
Prompts

Material density lbs/ftˆ3 (Enter for none): *enter a material density in lbs per cubic foot, or press Enter for none*

Ignore Zero Elevations [<Yes>/No]?

Select stockpile entities and perimeter.
Select objects: *pick the objects that define the stockpile and the 3D polyline perimeter*

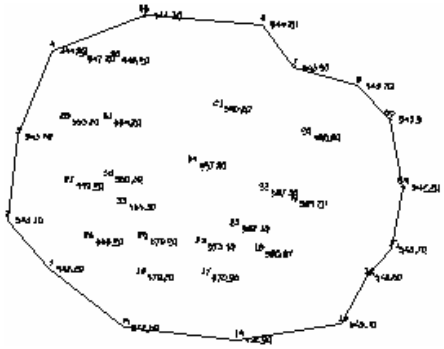
Select stockpile perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

Sample volume report

```
Volume report
Lower left grid corner: 15965.45,12657.05
Upper right grid corner: 16269.40,12906.29
X grid resolution: 50, Y grid resolution: 50
X grid cell size: 6.08, Y grid cell size: 4.98
Stockpile volume: 1191674.87825 cubic ft, 44136.107 cubic yards
```



Stockpile defined by points and a 3D polyline perimeter
Window these objects to obtain the volume report

Keyboard Command: stockvol
Prerequisite: Data representing the stockpile surface and a 3D polyline representing the perimeter of the stockpile.

Calculate Pond/Pit Volume

Function

This command is a customized and simplified method for calculating volumes in a situation in which the entire volume to be calculated is below the perimeter elevation, such as in the case of a pond or pit. The complimentary command, *Calculate Stockpile Volume*, is for the opposite situation, in which the entire volume to be calculated is above the elevation of the perimeter.

The program internally computes base and final grid surfaces from drawing geometry. The base surface is calculated from a 3D polyline representing the perimeter of the area being analyzed. If that 3D polyline is drawn on the PERIMETER layer, the command will automatically detect and use it. If no 3D polyline is found on that layer, you have an opportunity to manually select another 3D polyline to use. The 3D polyline perimeter can be drawn with the Draw 3D Polyline Perimeter command before using this routine.

The 3D polyline perimeter is also used as the inclusion perimeter for the volume calculation.

Additional 3D polylines can also be specified to more precisely define the base surface. These must be on the BASE_BREAKLINE layer to be used for this purpose. These can be generated by the *Draw 3DPoly Base Breakline* routine.

The final surface is calculated from all of the other selected drawing entities such as points, line, inserts, and polylines, along with the perimeter polyline, but not including the BASE_BREAKLINE polylines.

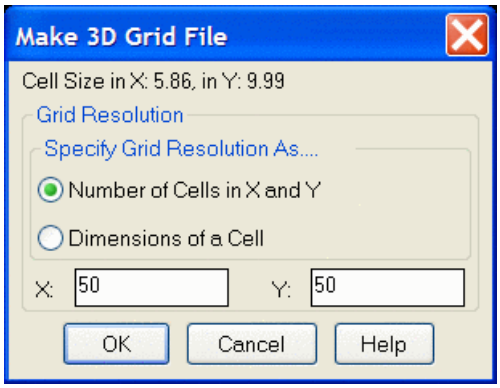
You have the option of setting the resolution of the grids.

Prompts

Ignore Zero Elevations [<Yes>/No]?

Select Pond/Pit entities and perimeter.
Select objects: *pick the objects that define the surface and the 3D polyline perimeter*

Select Pond/Pit perimeter polyline:



Make Grid File dialog Set the resolution and then click OK.

Keyboard Command: pitvol
Prerequisite: Data representing the pond/pit surface and a 3D polyline representing the perimeter of the pond/pit.

Calculate Total Volumes

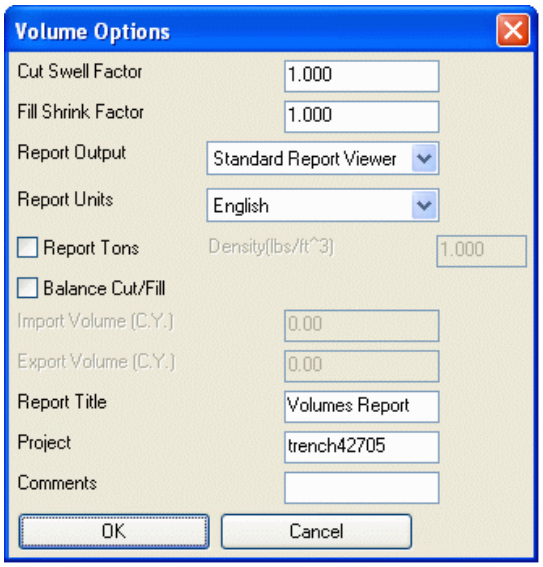
Function

Use this command to report total volume calculations within the site boundary polyline. The report includes the cut and fill quantities as well as the strata and topsoil quantities if the site has strata and topsoil defined. Besides reporting the total quantities for the site boundary, Area Of Interest polylines can be used to report quantities within named perimeters.

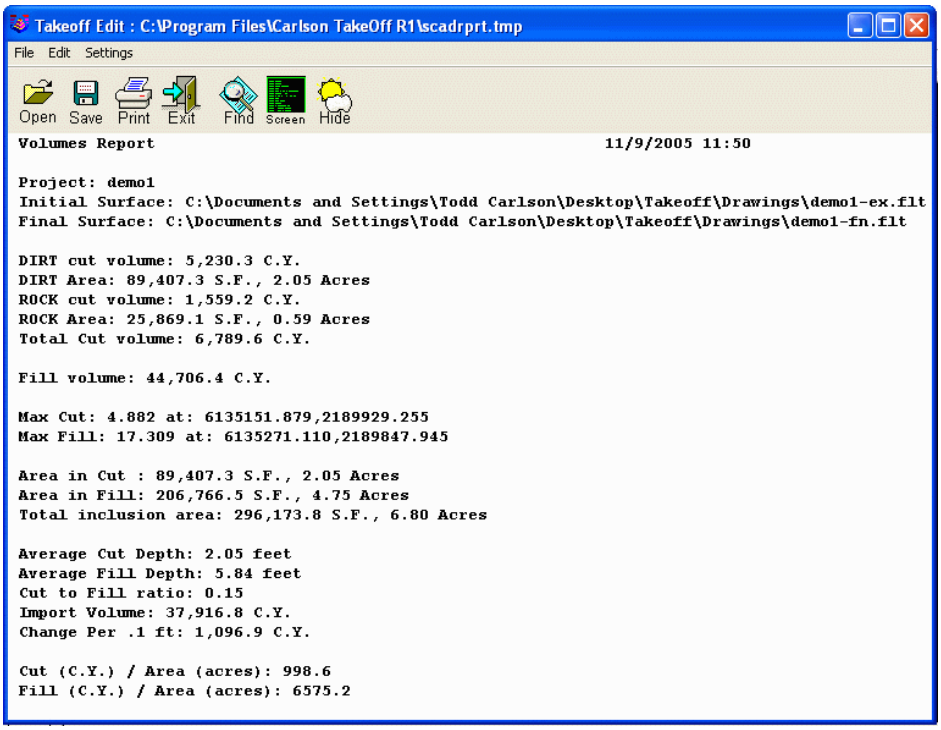
Before running this command, the exisiting and design surfaces must be created and the boundary polyline must be assigned. Also, the strata surfaces, topsoil and Area Of Interest polylines need to be set before this command if those features are to be reported.

The Volume Options dialog box shown here offers options for the final report. Here you can select four different types of reports: Standard Report Viewer, Custom Report Formatter, Expanded Auto Format, and Compressed Auto Format. The Cut Swell Factor is multiplied by the cut volume and the Fill Shrink Factor is multiplied by the fill volume. The Report Units setting chooses between English and Metric quantities for the report. In Drawing Setup in Takeoff, you set the drawing units as either English or Metric. The Report Units will default to match the drawing units but you can change the Report Units to the other mode and the program will apply the conversion between English and Metric for the report. So you can have a drawing in English units and create a report with Metric quantities.

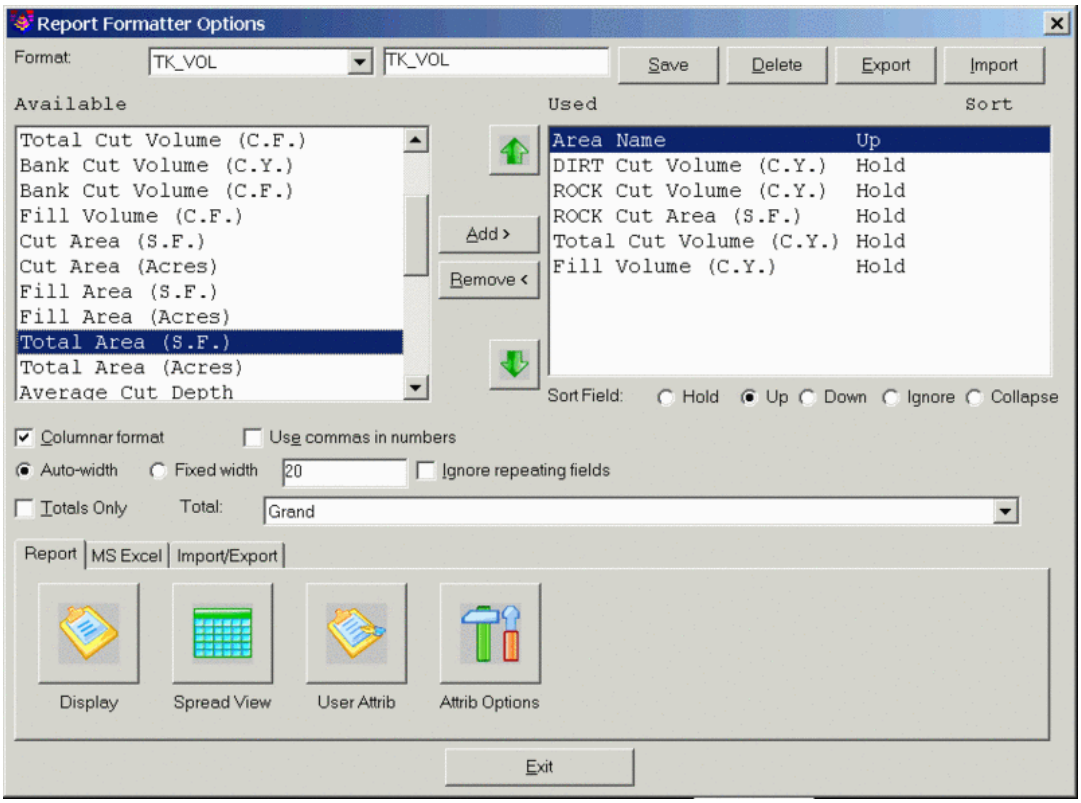
Note: As the quantities are calculated within each area, the area is hatched with a solid fill as a visual verification that the right area is being processed.



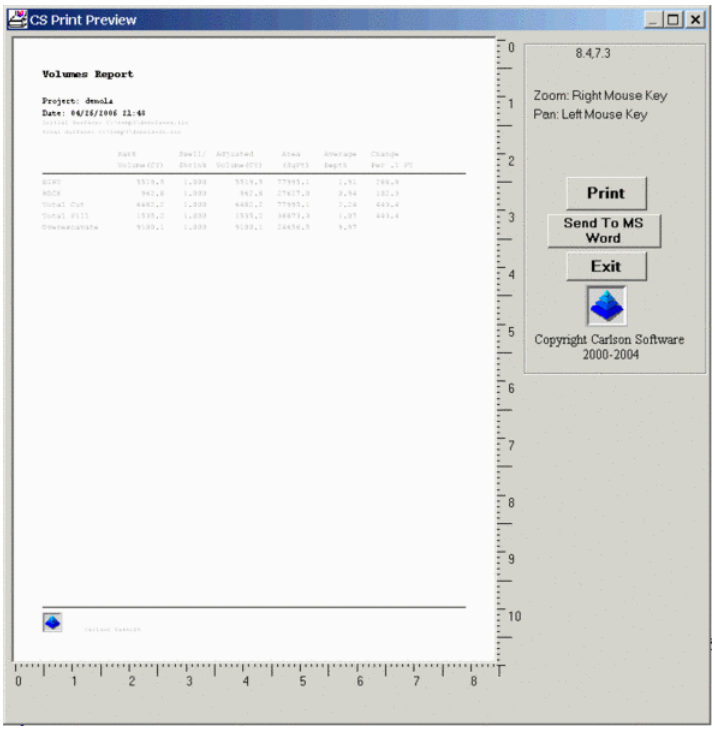
Shown here is an example of a Standard Report Viewer.


Takeoff Edit : C:\Program Files\Carlson TakeOff R1\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 11/9/2005 11:50
Project: demo1
Initial Surface: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo1-ex.flt
Final Surface: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo1-fn.flt
DIRT cut volume: 5,230.3 C.Y.
DIRT Area: 89,407.3 S.F., 2.05 Acres
ROCK cut volume: 1,559.2 C.Y.
ROCK Area: 25,869.1 S.F., 0.59 Acres
Total Cut volume: 6,789.6 C.Y.
Fill volume: 44,706.4 C.Y.
Max Cut: 4.882 at: 6135151.879,2189929.255
Max Fill: 17.309 at: 6135271.110,2189847.945
Area in Cut : 89,407.3 S.F., 2.05 Acres
Area in Fill: 206,766.5 S.F., 4.75 Acres
Total inclusion area: 296,173.8 S.F., 6.80 Acres
Average Cut Depth: 2.05 feet
Average Fill Depth: 5.84 feet
Cut to Fill ratio: 0.15
Import Volume: 37,916.8 C.Y.
Change Per .1 ft: 1,096.9 C.Y.
Cut (C.Y.) / Area (acres): 998.6
Fill (C.Y.) / Area (acres): 6575.2

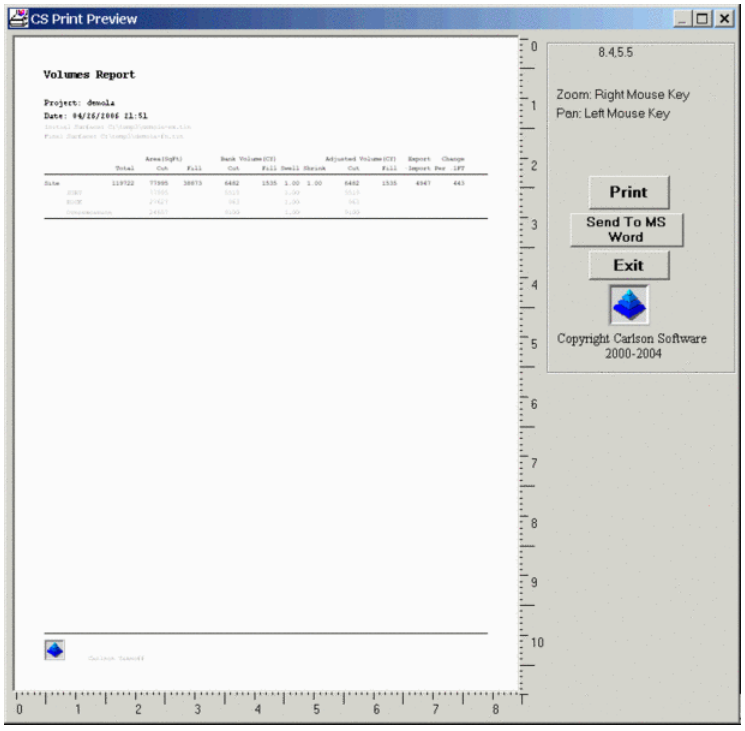
Use Customs Report Formatter to customize or "user define" the reporting options. The Report Formatter Options dialog box shown here offers a variety of output options including Excel. You can choose the fields to report from the Available list and set their report order under the Used list.



The Expanded Auto Format is shown in this report preview. You can also send the report to MS Word in this command.

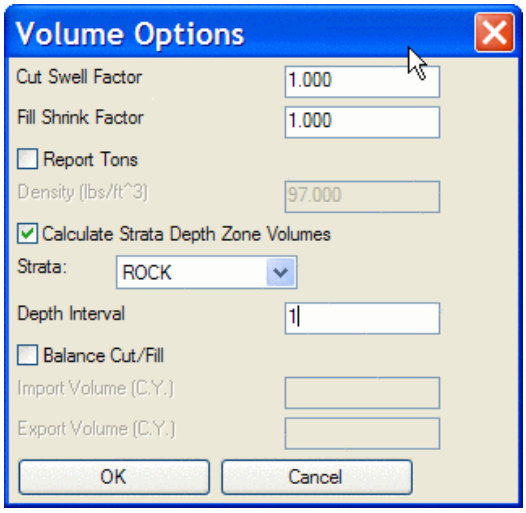


The Compressed Auto Format puts all the area information on one row.

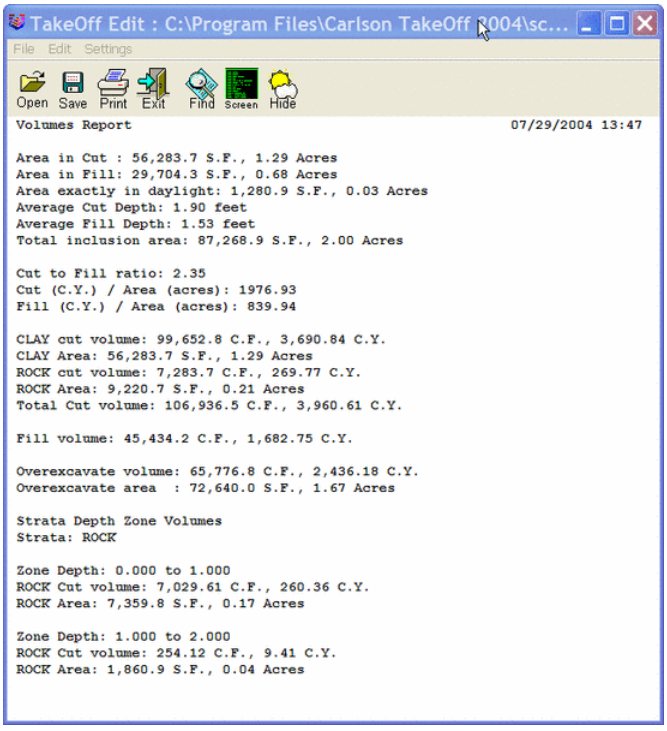


If drillholes have been located on the drawing and strata types and depths have been defined, a calculate

Strata Depth Zones Volume option
becomes available as shown here.

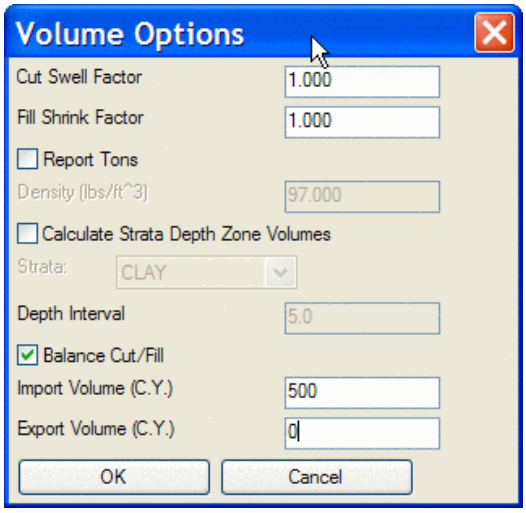


Shown here is an example of the report if strata depth intervals have been defined.

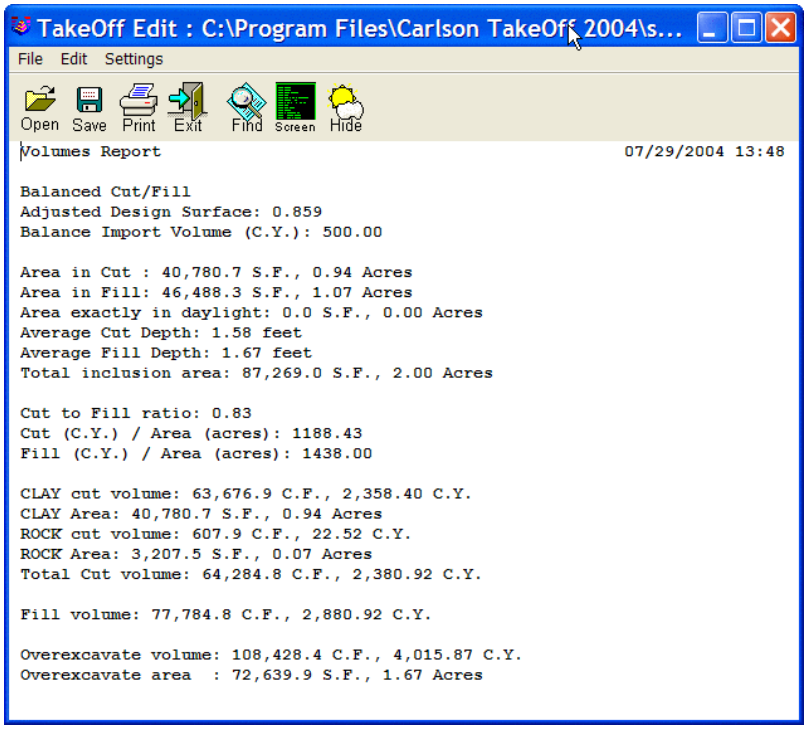

TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\sc...
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 07/29/2004 13:47
Area in Cut : 56,283.7 S.F., 1.29 Acres
Area in Fill: 29,704.3 S.F., 0.68 Acres
Area exactly in daylight: 1,280.9 S.F., 0.03 Acres
Average Cut Depth: 1.90 feet
Average Fill Depth: 1.53 feet
Total inclusion area: 87,268.9 S.F., 2.00 Acres
Cut to Fill ratio: 2.35
Cut (C.Y.) / Area (acres): 1976.93
Fill (C.Y.) / Area (acres): 839.94
CLAY cut volume: 99,652.8 C.F., 3,690.84 C.Y.
CLAY Area: 56,283.7 S.F., 1.29 Acres
ROCK cut volume: 7,283.7 C.F., 269.77 C.Y.
ROCK Area: 9,220.7 S.F., 0.21 Acres
Total Cut volume: 106,936.5 C.F., 3,960.61 C.Y.
Fill volume: 45,434.2 C.F., 1,682.75 C.Y.
Overexcavate volume: 65,776.8 C.F., 2,436.18 C.Y.
Overexcavate area : 72,640.0 S.F., 1.67 Acres
Strata Depth Zone Volumes
Strata: ROCK
Zone Depth: 0.000 to 1.000
ROCK Cut volume: 7,029.61 C.F., 260.36 C.Y.
ROCK Area: 7,359.8 S.F., 0.17 Acres
Zone Depth: 1.000 to 2.000
ROCK Cut volume: 254.12 C.F., 9.41 C.Y.
ROCK Area: 1,860.9 S.F., 0.04 Acres

The Balance Cut/Fill option shown here allows an import or export volume in cubic yards option. Use these

options if waste material is available or needed elsewhere. If this option is used the resulting report indicates the vertical movement of the site needed to satisfy the balance option.



Shown here is a report with a 500 CY importation of material and suggests that the site be vertically raised 0.859 feet.


TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\s...
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes Report 07/29/2004 13:48
Balanced Cut/Fill
Adjusted Design Surface: 0.859
Balance Import Volume (C.Y.): 500.00
Area in Cut : 40,780.7 S.F., 0.94 Acres
Area in Fill: 46,488.3 S.F., 1.07 Acres
Area exactly in daylight: 0.0 S.F., 0.00 Acres
Average Cut Depth: 1.58 feet
Average Fill Depth: 1.67 feet
Total inclusion area: 87,269.0 S.F., 2.00 Acres
Cut to Fill ratio: 0.83
Cut (C.Y.) / Area (acres): 1188.43
Fill (C.Y.) / Area (acres): 1438.00
CLAY cut volume: 63,676.9 C.F., 2,358.40 C.Y.
CLAY Area: 40,780.7 S.F., 0.94 Acres
ROCK cut volume: 607.9 C.F., 22.52 C.Y.
ROCK Area: 3,207.5 S.F., 0.07 Acres
Total Cut volume: 64,284.8 C.F., 2,380.92 C.Y.
Fill volume: 77,784.8 C.F., 2,880.92 C.Y.
Overexcavate volume: 108,428.4 C.F., 4,015.87 C.Y.
Overexcavate area : 72,639.9 S.F., 1.67 Acres

If the adjusted surface is satisfactory, Carlson Takeoff offers the option to save the adjusted surface as shown here in the Balance Cut/Fill dialog box.



Prerequisite: Existing and design surfaces and a boundary polyline
Keyboard Command: tin_volume

Calculate Volumes Inside Perimeter

Function

Use this command to create volume reports inside the selected closed perimeter polyline. The same reporting options are available for this command as are for the Calculate Total Volumes command.

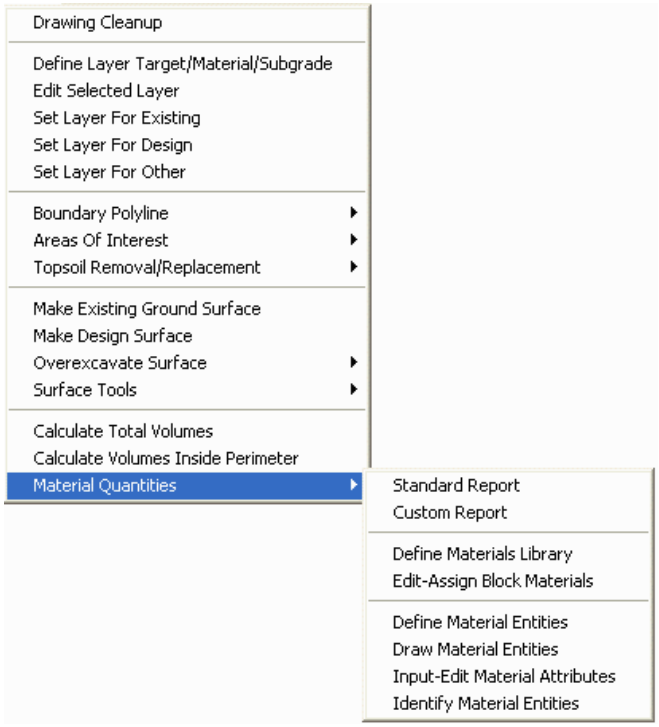
Prerequisite: Existing and Design surfaces and a closed perimeter polyline
Keyboard Command: tin_volume2

Material Quantities

Function

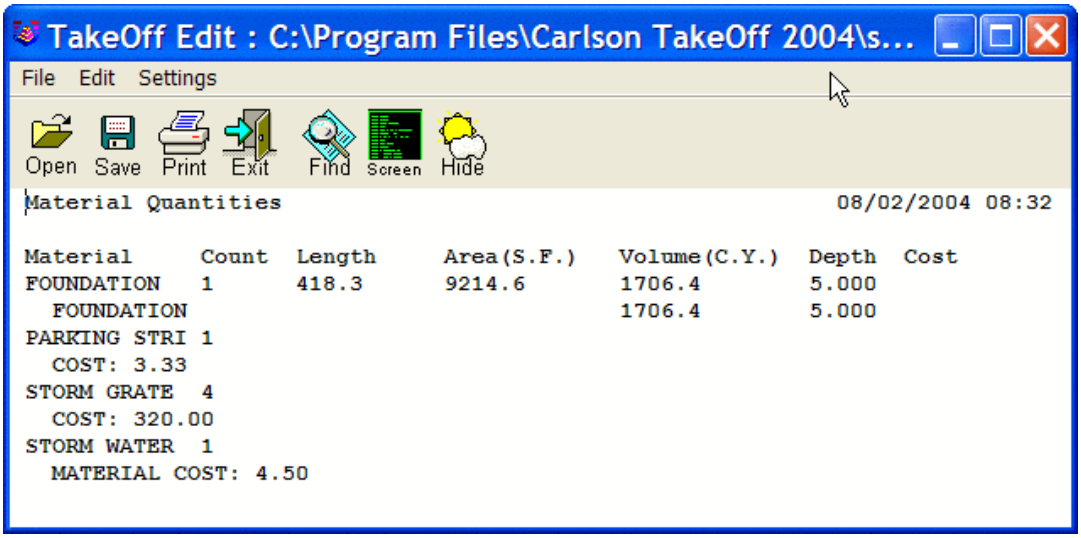
The Material Quantities flyout shown here offers many options for quantity reporting including the option for user defined attributes. Entities with attributes can be drawn, edited, and identified. Standard and custom report options are also available.

Material Quantities are counted from the entities in the drawing. Several entity properties can be reported including entity count, length, area and volume. Also user-defined attributes can be assigned to the entities and reported. The type of material for each entity is determined by the layer for the entity. In the Define Layer Target/Material/Subgrade, you can assign the material types by layer.



Standard Report

Use this command to display all or a selected set of material quantities and user-defined information with the standard Carlson Takeoff report format shown here.

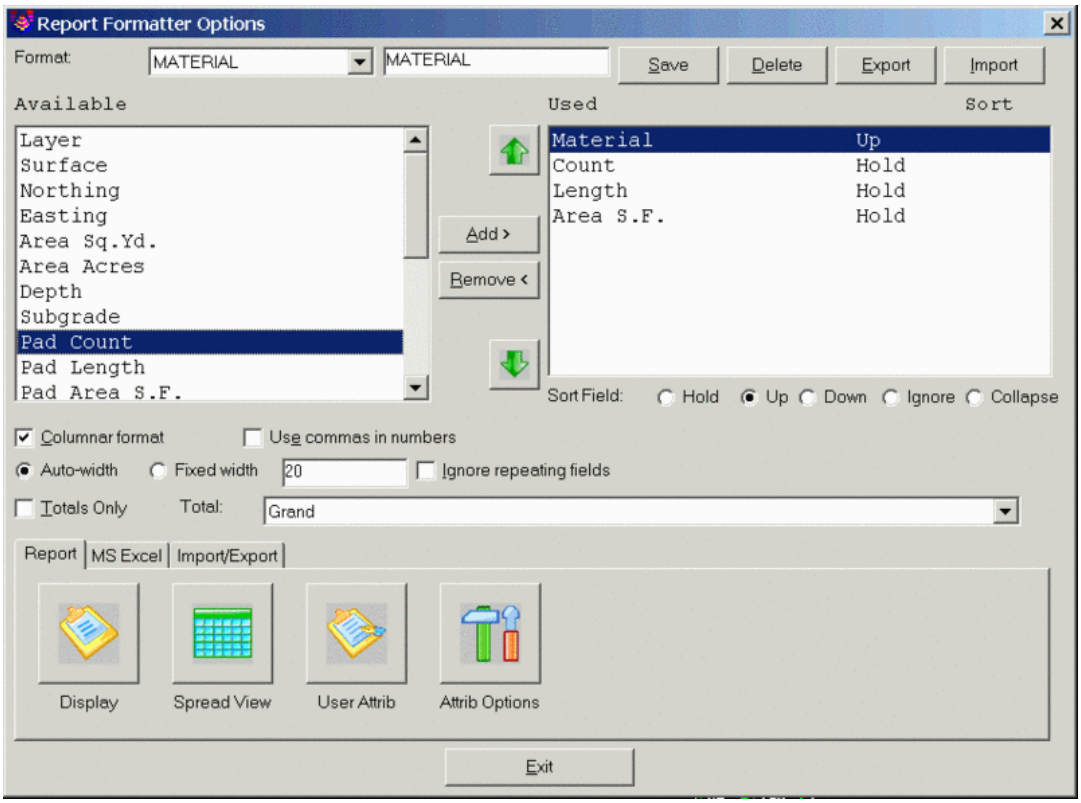

TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\s...
File Edit Settings
Open Save Print Exit Find Screen Hide
Material Quantities 08/02/2004 08:32
Material Count Length Area(S.F.) Volume(C.Y.) Depth Cost
FOUNDATION 1 418.3 9214.6 1706.4 5.000
FOUNDATION 1706.4 5.000
PARKING STRI 1
COST: 3.33
STORM GRATE 4
COST: 320.00
STORM WATER 1
MATERIAL COST: 4.50

Prerequisite: Defined materials

Keyboard Command: materials_report2

Custom Report

Use this command to customize or "user define" the reporting options. This command first prompts whether to report quantities for all the entities in the drawing or selected entities. Then if the drawing contains Areas Of Interest polylines, there is an option report quantities by these areas which adds the area name to the available report fields to allow sorting and grouping by area name. The Report Formatter Options dialog box shown here offers a variety of output options. You can choose the fields to report from the Available list and put them in report order under the Used list.



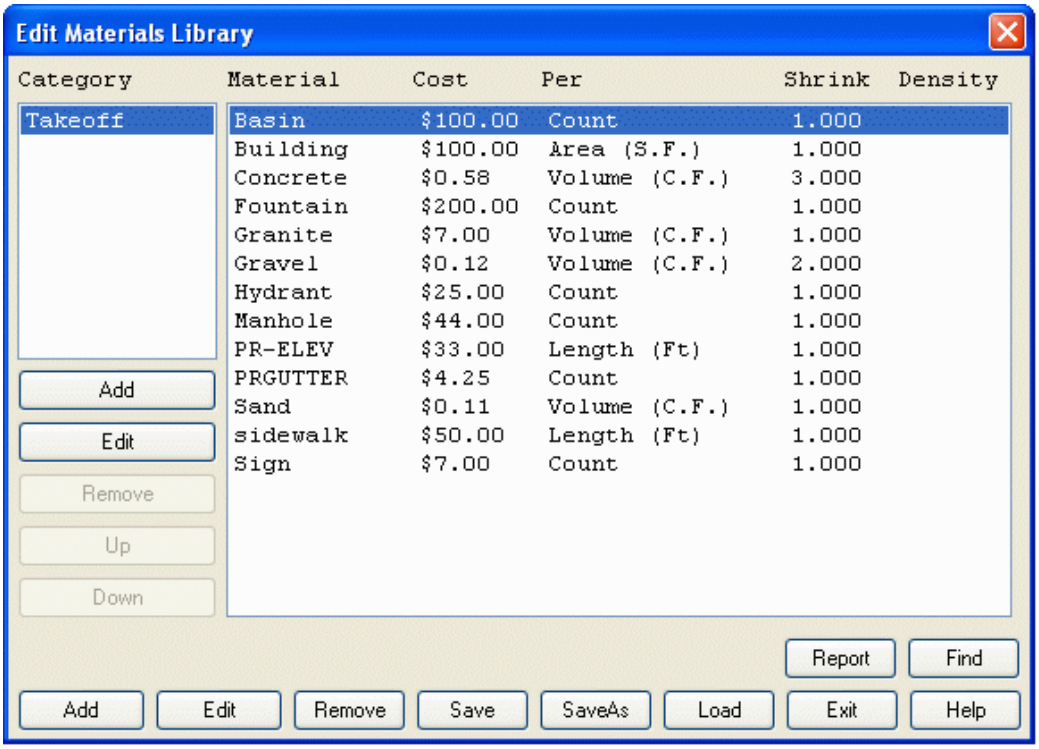
Selecting the Display option shows the report in the standard Carlson Takeoff report viewer. Reports can be exported to an Excel spreadsheet as well.

Prerequisite: Defined materials
Keyboard Command: materials_report

Define Materials Library

Define Materials Library allows you to Add, Remove, Load, Save, and Report a list of material costs. Costs can be set per area, count, volume, ton, or length by using the Edit function at the bottom of the dialog. The left side of the

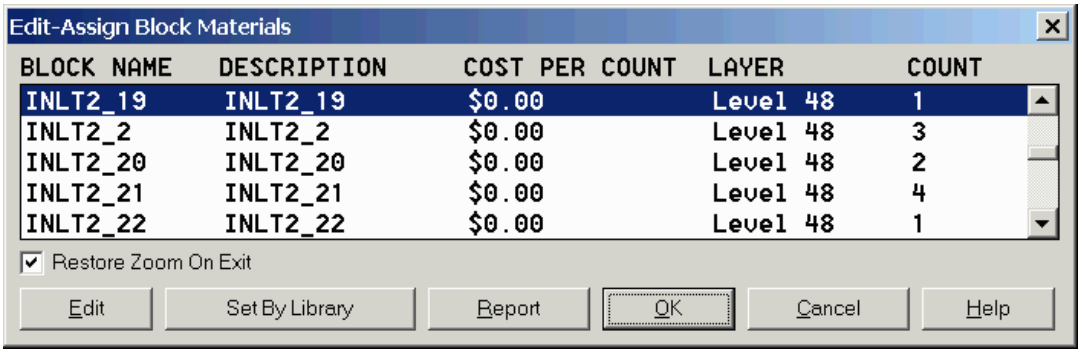
dialog can be used to set categories for different material costs.



Prerequisite: pricing for materials
Keyboard Command: define_tk_materials

Edit-Assign Block Materials

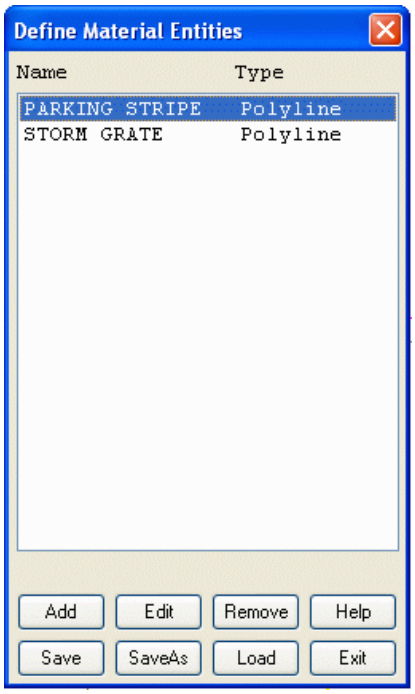
This command scans the current drawing to find and report block/symbol names and their count. For example, when the drawing contains different symbols for different types of utilities, this command identifies each type of symbol and the number. From this command, you can set the Description and Cost of the block by using the Edit buttom. You can also set the Description and Cost by predefined Materials by using the Set By Library button. When a block name is highlighted from the list, the drawing is zoomed to the location of one of those blocks so that you can see what it looks like. To Report these materials as part of the Standard Report, check on Include Materials Quantities Report in the Edit Materials dialog of the block layer found in the Define Layer Target/Material/Subgrade command. You can also just click on the Report button for a simple report.



Prerequisite: Blocks
Keyboard Command: edit_all_blocks

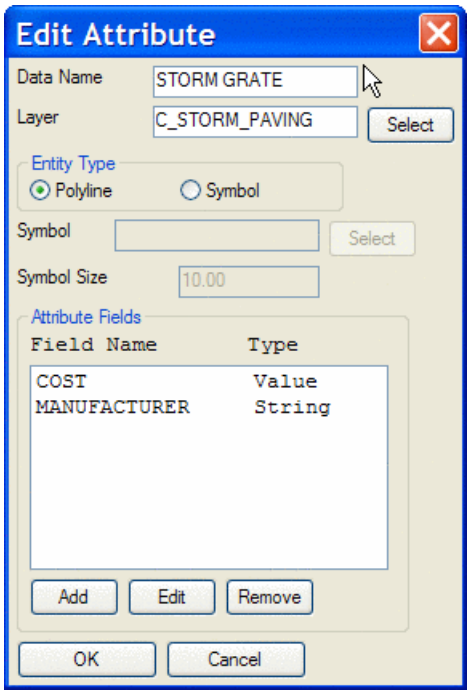
Define Material Attributes

Use this command to define all the material attributes that will be assigned to objects in the drawing for reporting purposes. The Define Attribute dialog box shown here allows the user to "Add", "Edit", or "Remove" attributes and save the definitions for later use. Simply "Load" a saved attribute definition file with the "tkd" extension for future use.

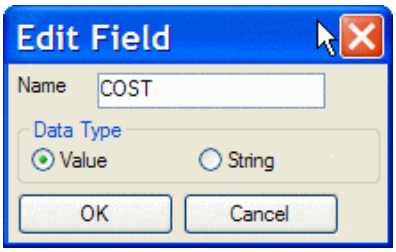


Selecting the Add or Edit options produce the edit attribute dialog box shown here. Use this command to define the Data name and the layer the objects currently reside on and the layer that future objects will be drawn on.

Two entity types can be used, polyline data or point data. If the symbol option is selected the user has the option of which symbol will represent the object. Attribute fields must be defined for material reporting.



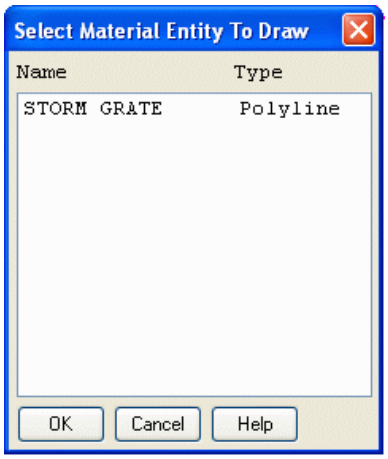
Selecting the Add or Edit button on the Edit Attribute dialog box brings up the Edit Field dialog box shown here. Use this dialog to define the field name and type. If the Value option is selected, only numeric values will be allowed when prompted. If the String option is selected, the user will have the ability to type in a text message when prompted.



Prerequisite: attributes
Keyboard Command: define_tk_data

Draw Materials Entities

Use this command to apply attribute data to objects as you draw or digitize them. Select the predefined attribute type to draw from the list available in the Select Attribute to Draw dialog box shown here.

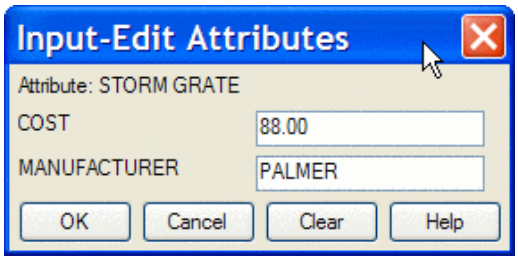


The command line will prompt the user to pick the points of the desired location of the object and allow the attribute data fields to be filled out upon completion or each "enter".

Prerequisite: defined attributes
Keyboard Command: draw_tk_data

Input-Edit Material Attributes

Use this command to assign predefined attribute information to an object already existing in the drawing. The command line prompt will require the user to select the object that attribute information is to be applied, and offer the Input-Edit Attribute dialog box shown here. This dialog box will display all predefined fields for that particular attribute type.



Prerequisite: predefined attribute information
Keyboard Command: edit_tk_data

Identify Materials Entities

Use this command to display all the objects that have attribute data assignments. The user will have the options of selecting the objects by picking them individually or by searching the entire drawing database. The objects that have attribute information assignments will "highlight" on the screen and the command line will display the attribute information.

Prerequisite: attributes
Keyboard Command: id_tk_data

Edit Menu 5

Undo

Function

This command allows you to reverse the effect of previously issued commands.

Prerequisite: None

Keyboard Command: U

Redo

Function

This command allows you to reverse the effects of the previous UNDO command.

Prerequisite: None

Keyboard Command: REDO

Erase, Select

Function

This command allows you to remove objects from a drawing.

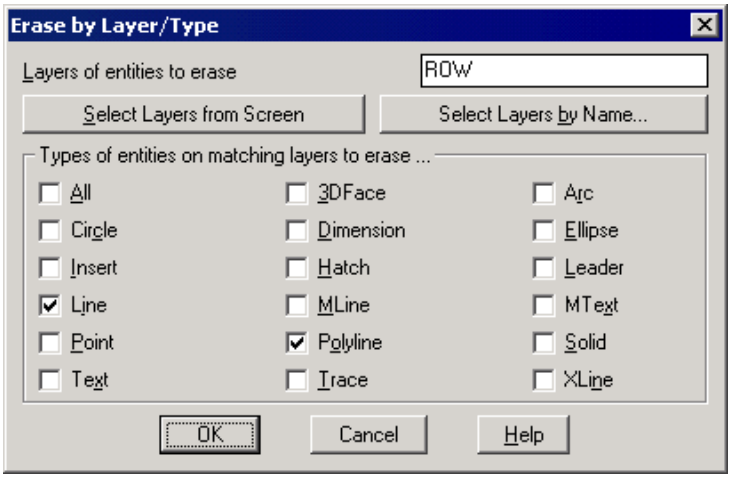
Prerequisite: None

Keyboard Command: ERASE, E

Erase by Layer

Function

This command erases all the entities on the specified layers and deletes the layers from the drawing.



1 In the Erase by Layer/Type dialog box, you can enter the name of the layer to be deleted, or you can specify a layer to delete by picking an entity on that layer.

- **Layers of entities to erase**: This field displays the layer names to be erased. You also have the option of typing layer names in this field.
- **Select Layers from Screen**: This option allows you to select entities on the layers you want deleted.
- **Select Layers by Name**: This option allows you to choose a layer name from the list of layers in the drawing.

2 Under Types of entities on matching layers to erase, you specify which types of entities to erase.

You can clear All and specify which entity types to include in the selection set. For instance, if you have both linework and points on the same layer, and you want to erase only the linework, you can clear All and select Line and Polyline.

Prerequisite: None

Keyboard Command: LDEL

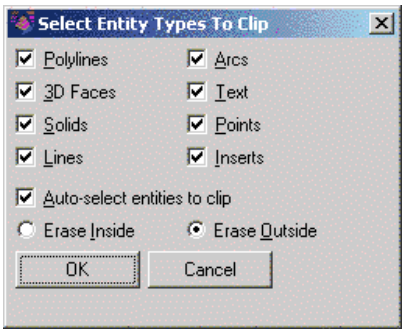
Erase by Closed Polyline

Function

This tool is used to cleanup drawing geometry at the extents of a polyline boundary. It provides options to erase adjacent geometry as well as trim geometry crossing the fence of the polyline.

First select the boundary polyline, only one can be selected. Designate the desired options in the following dialog.

The top section allows you to toggle which object types should be affected by the operation. Note that some objects such as text and inserts cannot be trimmed.



In the middle of the dialog is a toggle that determines whether to prompt for objects to process. If you want to isolate the drawings contents to that of the selected polyline, turn this toggle on. Note that all geometry in the drawing is effected, even that outside the current viewport. Many users will prefer to turn this toggle off so they will be prompted for the geometry to manipulate.

The bottom row allows you to choose whether to erase all geometry on the inside or outside.

Prerequisite: Entities, and a closed polyline

Keyboard Command: erasepline

Erase Outside

Function

This command erases all the entities outside of a user specified window. This can be useful if you somehow place entities way outside your drawing limits and want to easily erase them.

Pull-Down Menu Location: Edit, Erase->

Prerequisite: Entities to erase

Keyboard Command: eraseout

Move

Function

This command allows you to displace objects a specified distance in a specified direction.

Prerequisite: None

Keyboard Command: MOVE, M

Standard Copy

Function

This command copies all objects you select to the Clipboard. You can paste the contents of the Clipboard into a document or drawing as an OLE object.

You can also use CTRL+C to run this command. If the cursor is in the drawing area, Carlson TakeOff copies the selected objects to the Clipboard. If the cursor is on the command line or in the text window, the program copies the selected text to the Clipboard.

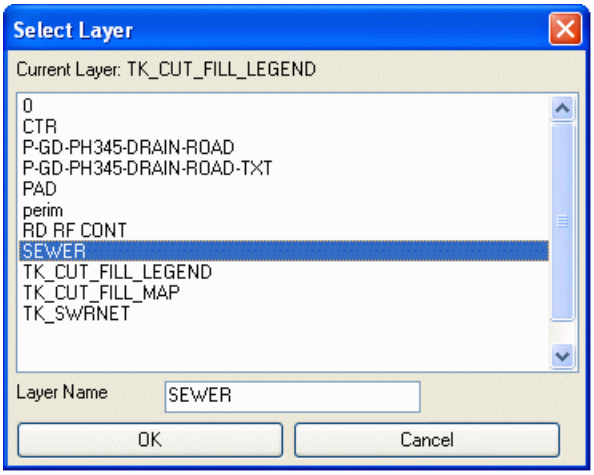
Prerequisite: None

Keyboard Command: COPY

Copy To Layer

Function

This command copies all objects you select to the Layer you decide.



Prerequisite: None

Keyboard Command: copy2layer

Standard Explode

Function

This command allows you to break a compound object into its component objects.

Results differ depending on the type of compound object you're exploding. The following is a list of objects that can be exploded and the results for each.

- **All Explodable Objects**: Produces object geometry that may look the same, but the color, linetype, and lineweight of the object may change.
- **Block**: Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a "*E" prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

- **2D and Lightweight Polyline**: Discards any associated width or tangent information.
- **Wide Polyline**: Places the resulting lines and arcs along the center of the polyline. TakeOff discards any associated width or tangent information.
- **3D Polyline**: Explodes into line segments. Any linetype assigned to the 3D polyline is applied to each resulting line segment.
- **Text Explode to Polylines**: Explodes polylines depending on the font used for various annotations, this can make the resulting polylines more efficient in terms of vertex count.
- **Leaders**: Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), Mtext, or tolerance objects, depending on the leader.

- **Mtext**: Explodes into text entities
- **Multiline**: Explodes into lines and arcs.
- **3D Solid**: Explodes planar surfaces into regions. Nonplanar surfaces explode into bodies.
- **Region**: Explodes into lines, arcs, or splines.
- **Body**: Explodes into a single-surface body (nonplanar surfaces), regions, or curves.
- **Polyface Mesh**: Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.
- **Circle Within a Nonuniformly Scaled Block**: Explodes a circle within a nonuniformly scaled block into ellipses.
- **Arc Within a Nonuniformly Scaled Block**: Explodes an arc within a nonuniformly scaled block into elliptical arcs.

Prerequisite: None

Keyboard Command: EXPLODE, X

Block Explode

Function

This command retains the values of attributes when a block is exploded. The standard AutoCAD *Explode* command changes the attribute values back to the attribute type. For example, using *Explode*, a Carlson point block would become PNTNO, PNTELEV, PNTDESC. *Block Explode* would keep the point attribute values such as 10, 1000.0, EP. The layer names of the exploded block attributes can be either the insert layer of the parent block or the original attribute layers from the block definition.

Prerequisite: A block to be exploded

Keyboard Command: explode2

Align

Function

Aligns objects with other objects in 2D and 3D

You use ALIGN to move, rotate, or scale objects into alignment with other objects. Add source points to the objects you want to align, and add destination points to the objects to which you want the source objects to align. You can add up to three pairs of source and destination points to align an object.

The first set of source and destination points defines the base point for the alignment. The second set of points defines the angle of rotation.

When you select three point pairs, you can move and rotate the selected objects in 3D to align with other objects.

If you use two source and destination points to perform a 3D alignment on nonperpendicular working planes, you get unpredictable results.

After you enter the points, Takeoff prompts you to scale the object. The program uses the distance between the first and second destination points as the reference length to which the object is scaled. Scaling is available only when you are aligning objects using two point pairs.

Prompts

1 Specify first source point: **pick point**

2 Specify first desitination point: **pick point**

3 Specify second source point: **pick point**

4 Specify second desitination point: **pick point**

5 Specify third source point or <continue>: **Press Enter**

6 Scale objects based on alignment points? [Yes/No] <N>: **Press Enter**

Prerequisite: None

Keyboard Command: ALIGN

2D Align

Function

This command will align (translate, rotate and scale) the selected objects using two pairs of source and destination control points. The difference between the first source point and first destination point determines the translation amount. The difference between the angle and distance from the first and second source points compared to the angle and distance from the first and second destination points determines the rotation and scale. The scale part of the alignment is optional. This 2D Align function is the same as the AutoCAD Align function except that this 2D Align function does not use elevations so that the alignment is always in 2D. The control points can be screen picked or entered by point numbers.

Prompts

Select entities to align.
Select objects: *pick entities to process*
First Source Point?
Pick point or point number: *pick a point*
First Destination Point?
Pick point or point number: *pick a point*
Second Source Point?
Pick point or point number: *pick a point*
Second Destination Point?
Pick point or point number: *pick a point*
Scale factor: 1.00434258
Scale objects based on alignment points [Yes/<No>]? *N*

Keyboard Command: scalign
Prerequisite: None

Trim

Function

This command allows you to trim objects at a cutting edge defined by other objects.

Prompts

1 Select cutting edges ...

Select objects: **pick entity**

2 Select object to trim or shift-select to extend or [Project/Edge/Undo]: **select entity to be trimmed**

- **Project**: You can project the object to be trimmed in order to trim objects that do not intersect.
- **Edge**: You can project the trimming edge in order to trim objects that do not intersect.
- **Undo**: This option allows you to undo the above projections.

Prerequisite: None

Keyboard Command: TRIM, TR

Scale

Function

This command allows you to enlarge or reduce selected objects equally in the X, Y, and Z directions.

Prompts

1 Select objects: **pick entities**

2 Specify base point: pick point on screen as reference

3 Specify scale factor or [Reference]: **scale to desired size**

Prerequisite: None

Keyboard Command: SC

Extend To Edge

Function

This command allows you to extend an object to meet another object.

Prompts

1 Select boundary edges ...

Select objects: **pick entity**

2 Select object to extend or shift-select to trim or [Project/Edge/Undo]: **pick entity**

You have the option of trimming or projecting objects and edges.

Prerequisite: None

Keyboard Command: EXTEND

Extend by Distance

Function

This command extends a line or polyline, or creates a new line or polyline from an existing one, using a specified distance. The new segment of the line or polyline is drawn from the current position in the direction indicated by the current position arrowhead.

You start by selecting an existing line or polyline. Initially, the current position is the closest vertex to the location where the line or polyline was selected. Extending from the endpoint of a polyline will add a new point to that polyline, while extending from any other point will create a new polyline.

There are two modes of operation: draw mode (D) and move mode (M). When you are in draw mode, extending will draw line or polyline segments. When you are in move mode, the current position arrowhead can be moved without drawing segments. The orientation of the current position arrowhead can be changed with the Right, Left, and Angle commands.

Here is a list of the Extend by Distance options:

Key Name Action

D	Draw mode	Actions draw or extend the line or polyline
M	Move mode	Actions only move the pointer
#	Number	Distance to draw or extend
R	Right rotate	Rotates clockwise 90 degrees
L	Left rotate	Rotates counterclockwise 90 degrees
E	Extend to edge	Extends to intersection with a selected line or polyline
T#	Total distance	Sets current segment to specified distance
A#	Angle change	Rotates pointer by specified number of degrees
A	Align	Rotates pointer to align with segment
B	Bearing	Sets pointer direction by bearing in format: Qdd.mmss with Q-quadrant, d-degrees, m-minutes, s-seconds (e.g. 130.1005 is NE 30 degrees, 10 minutes, and 5 seconds)
S	Switch	Reverses pointer direction
N	Next	Moves pointer to next point
P	Previous	Moves pointer to previous point
U	Undo	Undo the last Extend by Distance command
C	Close	Closes the polyline
O	Open	Opens the polyline
Help	Help	Displays this list

Prompts

1 Select line or pline to extend: **select line or polyline**

Pick the polyline near the place to extend

2 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **T50**

The line is extended to a total length of 50 units.

3 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **R**

The pointer is turned to the right.

4 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **75**

A polyline is drawn to 75 units.

5 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **B145.0000**

The pointer is turned toward NE at a bearing of 45 degrees.

6 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **50**

A polyline is drawn to 50 units.

7 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **Press Enter**

Press Enter to end the command.

Prerequisite: An existing line or polyline with at least one segment from which to start.

Keyboard Command: EXTENDER

Break by Closed Polyline

Function

This command will force a break where all lines and polylines cross the "break" polyline. In addition to breaking lines and polylines, you can also put the segments to new layers based on whether the segments are inside or outside the break polyline. Polylines with arc segments are not handled. To reconnect the broken lines and polylines, use the Join Nearest command.

Prompts

1 Select the clip edge polyline: **pick a closed polyline**

2 Select the polylines and lines to be clipped.

Select Objects: **pick the entities to break**

3 Specify layer names for Inside segments (Yes/<No>)? **Yes**

4 Enter a layer name for the Inside segments <0>: **Press Enter**

5 Specify layer names for Outside segments (Yes/<No>)? **Yes**

6 Enter a layer name for the Outside segments <0>: **Final**

Prerequisite: A closed polyline

Keyboard Command: CLIPLINE

Break At Selected Point

Function

This command allows you to break an object by selecting the object. Only one pick is necessary since TakeOff both selects the object and treats the selection point as the break point.

Prompts

1 Select object: select entity to break

Select an object to break

Prerequisite: None

Keyboard Command: BREAK

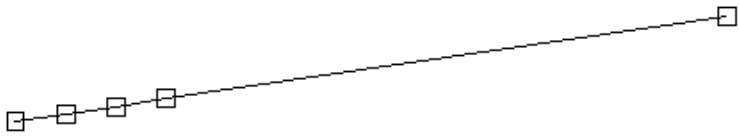
Break Polyline at Specified Distances

Function

This command allows you to pick a polyline and break it at a specified distances along the polyline. Following the prompts below, the beginning of the polyline in the illustration was broken into three 55-foot segments.

Prompts

Select polyline to break: *select polyline*
Total Distance: 779.429 This is the length of the polyline reported.
Distance Along Polyline For Break: *55.0*
Distance Along Polyline For Break (Enter to end): *110*
Distance Along Polyline For Break (Enter to end): *165*
Distance Along Polyline For Break (Enter to end): *press Enter*
3 polyline breaks created.



Keyboard Command: breakpl
Prerequisite: A polyline to break

Break at Intersection

Function

This command allows you to break a line, arc, or polyline at the intersection of another line, arc, or polyline. This command is often used in conjunction with the Area by Lines & Arcs command because many times, to get the correct area of a figure, you must break it from adjoining lines.

Prompts

1 Select Line, Arc, or Polyline to Break

Select object: **select entity to break**

2 [int on] Pick Intersection to break at: **pick intersection point**

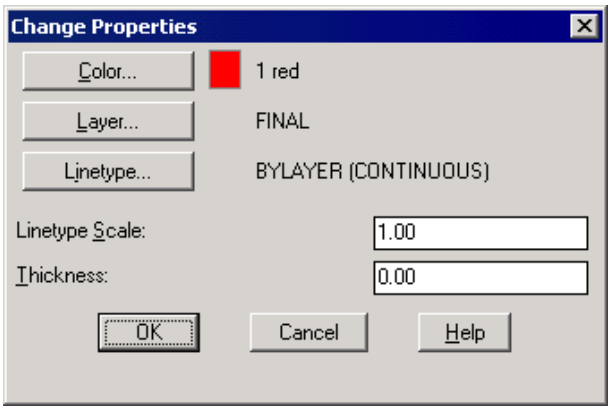
Prerequisite: None

Keyboard Command: BREAKAT

Change Properties

Function

This command allows you to change certain properties of existing objects.



1 In the Change Properties dialog box, you must choose the properties to modify.

- **Color**: This option allows you to change the color of the object.
- **Layer**: This option allows you to change the layer of the object.
- **Linetype**: This option allows you to change the linetype of the object.
- **Linetype Scale**: This option specifies the linetype scale factor for the new linetype.
- **Thickness**: This option specifies the distance to extrude the object above or below its elevation.

Note: The Properties command allows you to modify entity specific properties such as the radius of a circle or the height of a text entity.

Prerequisite: None

Keyboard Command: DDCHPROP

Rotate by Bearing

Function

This command allows you to move objects about a base point by a given bearing.

Prompts

1 Select entities to rotate.

Select objects: **pick entities**

2 Base pivot point?

Pick point or point number: **pick a point**

3 Reference Bearing point?

Pick point or point number: **pick a point**

4 Azimuth/<New Bearing (Qdd.mmss)>: **enter a bearing**

Prerequisite: None

Keyboard Command: BROT

Standard Rotate

Function

This command allows you to move objects about a base point using a point as a rotation reference.

Prompts

1 Select objects: **pick entities**

2 Specify base point: pick point on screen as reference

3 Specify rotation angle or [Reference]: **rotate to desired location**

Prerequisite: None

Keyboard Command: ROTATE

Edit Text

Function

This command allows you to edit text and attribute labels.

1 Select Text to Edit: **select the text**

You can modify text in provided text field.

Prerequisite: Text

Keyboard Command: EDITXT

Find_and_Replace_Text

Function

With this command, you can find, replace, select, or zoom to text contained in the current drawing.

Keyboard Command: find
Prerequisite: Text

Text EnlargeReduce

Function

This command allows you to scale text entities up or down in size. The command prompts for a scale multiplier and a selection set of text objects. If you want to enlarge the text, enter a value greater than one. If you want to reduce the text, enter a decimal fraction such as .5, which would reduce the text size by 50%. Use this command if you set up your drawing for one plotting scale and then decide to change to a new plotting scale. The Change Text Size command can alternatively be used to set the text size to a specific value.

Prompts

1 Scaling Multiplier: **enter value**

Select Text for Scaling.

2 Select objects: **select the text**

Prerequisite: Text

Keyboard Command: TXTENL

Text Explode To Polylines

Function

This command converts the selected text into polylines. This function is generally used when preparing a plan view file for Dozer 2000 before using the *Write Polyline File* command.

Prompts

Select text to be EXPLODED.

Select objects: *select the text*

1 text object(s) have been exploded to lines.

The line objects have been placed on layer 0.

Reading the selection set ...

Joining ...

Converting ...

Prerequisite: Text

Keyboard Command: textexp

Image Frame

Function

This command controls whether TakeOff displays the image frame or hides it from view.

Because you select an image by clicking its frame, setting the image frame to off prevents you from selecting an image.

Prompts

1 Enter image frame setting [ON/OFF] <current>: **enter an option or Press Enter**

- **On**: Displays image frames so you can select images.
- **Off**: Hides image frames so you cannot select images.

Prerequisite: None

Keyboard Command: IMAGEFRAME

Image Clip

Function

This command allows you to create new clipping boundaries for an image object.

Prompts

1 Select image to clip: select the edge of an image

2 Enter image clipping option [ON/OFF/Delete/New boundary] <New>: **enter an option or Press Enter**

The boundary you specify must be in a plane parallel to the image object.

- **On**: Turns on clipping and displays the image clipped to the previously defined boundary.
- **Off**: Turns off clipping and displays the entire image and frame. If you reclip the image while clipping is turned off, the program automatically turns clipping back on. The program prompts you to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.
- **Delete**: Removes a predefined clipping boundary and redisplays the full original image.
- **New Boundary**: Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the image boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the program interprets the point as the first corner of a rectangle.

3 Enter clipping type [Polygonal/Rectangular] <Rectangular>: **enter P or Press Enter**

- **Polygonal**: Uses specified points to define a polygonal boundary.

Specify first point: Specify a point

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Close/Undo]: **specify a point, or enter c or u**

You must specify at least three points to define a polygon.

If the image already has a clipping boundary defined, TakeOff displays the following prompt:

Delete old boundary? [No/Yes] <Yes>: **enter N or Press Enter**

If you choose Yes, the program redraws the entire image and the command continues; if you choose No, the command ends.

- **Rectangular**: Specifies a rectangular boundary by its opposite corners. TakeOff always draws the rectangle parallel to the edges of the image.

Specify first corner point: **specify a point**

Specify opposite corner point: **specify a point**

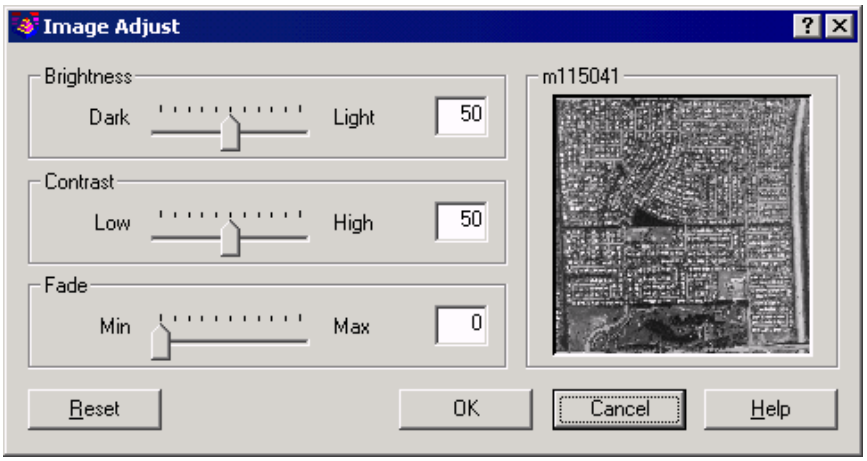
Prerequisite: None

Keyboard Command: IMAGECLIP

Image Adjust

Function

This command controls the display of the brightness, contrast, and fade values of images.



The Image Adjust dialog box controls how the image is displayed by adjusting the brightness, contrast, and fade settings of the selected image. Adjusting these values changes the display of the image but does not change the image file itself.

- **Brightness**: Controls the brightness, and indirectly the contrast, of the image. Values range from 0 through 100. The greater the value, the brighter the image and the more pixels that become white when you increase contrast. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Contrast**: Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Fade**: Controls the fading effect of the image. Values range from 0 through 100. The greater the value, the more the image blends with the current background color. A value of 100 blends the image completely into the background. Changing the screen background color causes the image to fade to the new color. In plotting, the

background color for fade is white. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Image Preview**: Displays a preview of the selected image. The preview image updates dynamically to reflect changes to the brightness, contrast, and fade settings.
- **Reset**: Resets values for brightness, contrast, and fade to default settings (50, 50, and 0, respectively).

Prerequisite: None

Keyboard Command: IMAGEADJUST

Remove Groups

Function

This command breaks up AutoCAD groups into more managable entities.

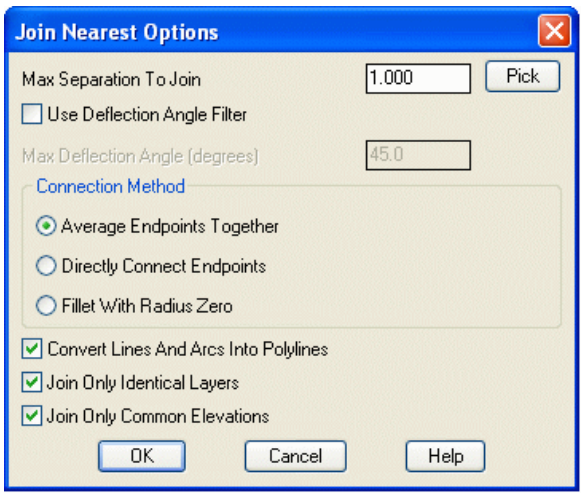
Prerequisite: Groups

Keyboard Command: rmgroup

Join Nearest

Function

This command joins lines or polylines together, and allows you to join lines that do not exactly meet. You specify the maximum distance to join, along with other options, in the dialog box shown below. You can join many entities at once.



1 First, specify the maximum separation distance parameter and the maximum deflection angle. Entities beyond this distance or angle will not join.

2 Under Connection Method you must determine how entities are connected.

- **Average Endpoints Together**: This option averages together the endpoints of the two entities when joined.
- **Directly Connect Endpoints**: This option directly connects the endpoints of the two entities with a polyline.
- **Fillet With Radius Zero**: This option will perform a tight fillet on the two entities.

3 In the Join Nearest Options dialog box you can choose to join only lines with common elevations or layers.

- **Convert lines into polylines**: The option will automatically convert any lines in the selection set into polylines.
- **Join only identical layers**: This option will join entities on the same layer.
- **Join only common elevations**: This option will join entities with identical elevations.

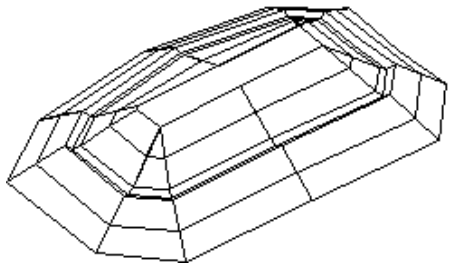
Prerequisite: Lines or Polylines to be joined.

Keyboard Command: NEARJOIN

Offset Polyline

Function

This command allows you to offset a 3D polyline entity in both the horizontal and vertical directions. There are three offset methods. The Interval method applies one horizontal and one vertical offset to all the vertices of the polyline. The Constant method uses a horizontal offset and sets the elevation of the polyline to one constant elevation. The Variable method allows you to specify each horizontal and vertical offset individually, either by polyline segment or for each point. The vertical offset can be specified by actual vertical distance, percent slope, or slope ratio.



Prompts

1 Enter the offset method (<Interval>/Constant/Variable): **Press Enter**

2 Vertical/<Horizontal offset amount>: **15**

3 Percent/Ratio/Vertical offset amount <0>: **10**

4 Select a polyline to offset (Enter for none): **select a 3D polyline**

5 Select side to offset: **pick point**

Select a point on the graphics screen in the direction of the desired offset.

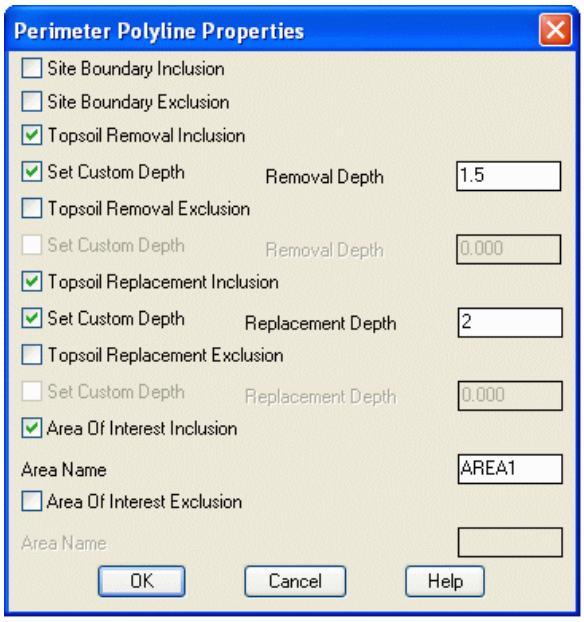
Prerequisite: 3D polylines

Keyboard Command: OFFSET3D

Perimeter Polylines Properties

Function

This command allows you to control the properties of any perimeter polyline (Note: Perimeter polylines also have to be closed polylines). Select a polyline and the following dialog appears. Here you can define the functionality of the polyline in regards to, the Site Boundry, Areas of Interest, and Topsoil Removal/Replacement. These properties can also be set separately using the Boundry Polyline, Areas of Interest, and Topsoil Removal/Replacement commands found under the Tools menu of Carlson Takeoff.



Prerequisite: a polyline

Keyboard Command: perim_prop

3D Entity to 2D

Function

This command changes a 3D Arc, Line, or Polyline to a 2D Line or Polyline, resulting in an entity whose endpoints or vertices all have the same Z coordinate. When you select a 3D polyline with all vertices with the same elevation, you can use an option that converts it to a 2D polyline with this elevation. If the vertices of the selected polyline have different elevations, you must enter an elevation.

Prompts

1 New Elevation <0.00>: **Press Enter**

2 Select Lines, Arcs or Polylines for elevation change.

Select objects: **pick a 3D entity**

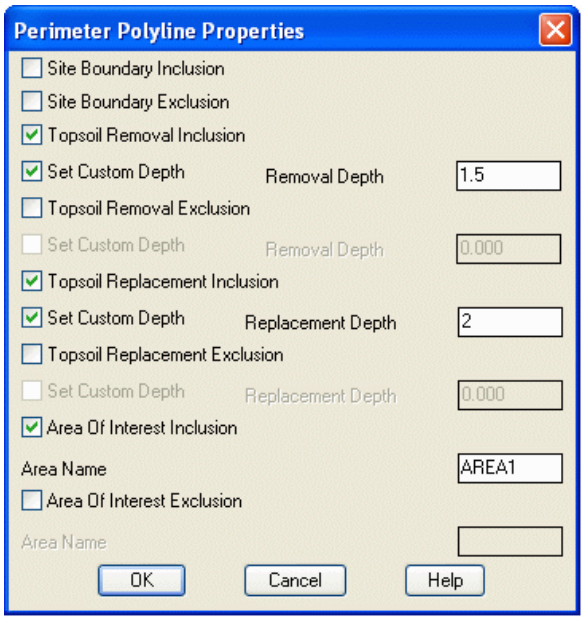
Prerequisite: 3D entities

Keyboard Command: 3DTO2D

Perimeter Polylines Properties

Function

This command allows you to control the properties of any perimeter polyline (Note: Perimeter polylines also have to be closed polylines). Select a polyline and the following dialog appears. Here you can define the functionality of the polyline in regards to, the Site Boundry, Areas of Interest, and Topsoil Removal/Replacement. These properties can also be set separately using the Boundry Polyline, Areas of Interest, and Topsoil Removal/Replacement commands found under the Tools menu of Carlson Takeoff.



Prerequisite: a polyline

Keyboard Command: perim_prop

Entities to Polylines

Function

This command converts selected lines, arcs, circles, 3Dfaces, and solids into individual polylines. You may use Join Nearest to convert adjoining lines and arcs into continuous polylines.

Prompts

1 Select lines, arcs, circles, 3Dfaces and solids to convert.

Select objects: **pick entities**

Prerequisite: Lines, arcs or other entities to convert.

Keyboard Command: TOPLINE

Reverse Polyline

Function

This command reverses the order of the line and/or arc segments of a polyline. This can be used in conjunction with the commands such as Station Polyline/Centerline or Profile from Surface Entities, since the polyline must be plotted in the direction of increasing stations. If it is more convenient to draft a polyline in one direction you may do so and then use the Reverse Polyline command to change its order. This command can also be used to reverse a 3D Polyline Breakline or a 3D Pad Template. Temporary arrows are drawn along the polyline to graphically show the new polyline direction.

Prompts

1 Select the Polyline to Reverse: **pick point on polyline**

Prerequisite: A polyline

Keyboard Command: REVPLINE

Reduce Polyline Vertices

Function

This command removes points from a polyline without significantly changing the polyline. The offset cutoff is the maximum distance that the polyline can move when you remove a point. For example, in a polyline with three points in a straight line, the middle point can be removed without changing the polyline.

Prompts

1 Enter the offset cutoff <0.1>: **.5**

2 Select polylines to reduce.

Select objects: **pick polylines**

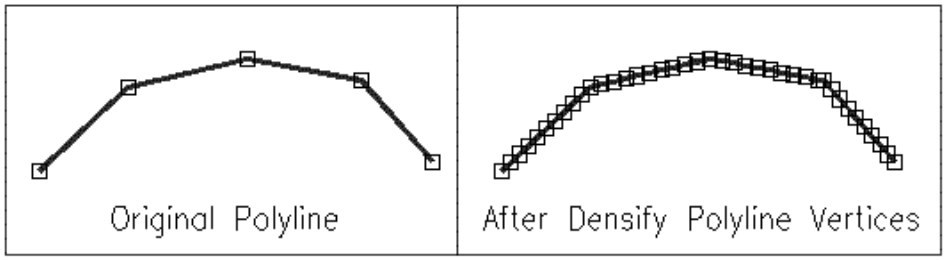
Prerequisite: A polyline

Keyboard Command: REDUCE

Densify Polyline Vertices

Function

This command adds vertices to the selected polylines at the specified interval. These points are interpolated between existing points in the polyline. This command is the opposite of Reduce Polyline Vertices.



Prompts

Select polylines to densify.

Select objects: *select polylines*

Point interval <10.0>: *Press Enter*

Testing Entity> 1

Added 17 points to 1 polyline.

Prerequisite: A polyline.

Keyboard Command: densepl

Draw Polyline Blips

Function

This command will draw temporary markers, "blips", at each polyline vertex. This allows you to identify the actual location of each vertex. The Blips are temporary. Any change to the viewport (pan, zoom, regen) will make the blips disappear. In later versions of AutoCAD, you can also click on the polyline to activate the grips which will remain visible during and after viewport changes. See illustration under *Densify Polyline Vertices.*

Prompts

Select polylines to draw blips.

Select objects: *select polyline(s)*

Prerequisite: A polyline.

Keyboard Command: plblip

Set Polyline Origin

Function

This command sets the starting vertex of a closed polyline. Select the polyline, then pick near the point you want to set as the starting point.

Prompts

1 Select Polyline: **pick a polyline**

2 Pick Near New Origin Point: **pick a point on the polyline** to be the starting point

Prerequisite: A closed polyline

Keyboard Command: PLCHGORG

Add Intersection Points

Function

This command allows you to add points to lines or polylines at intersections with other lines or polylines. This can be useful for commands such as Auto Annotate. For example, use Add Intersection Points to add points to a boundary polyline where the lot lines intersect. Then use Auto Annotate to label the boundary distance along each lot. This command does not add intersection points on arcs.

Prompts

1 Select lines and polylines to process.

Select objects: **pick lines or polylines**

Prerequisite: Polylines or lines

Keyboard Command: ADDINT

Add Polyline Vertex

Function

This command allows you to add points to a polyline. Select the polyline to modify, then pick or enter the coordinates for the new point. The new point is inserted into the polyline at the nearest polyline segment. On a 3D polyline, the elevation of the new vertex is calculated for you.

Prompts

1 Select polyline to add to: **pick a polyline**

2 Pick or enter point to add: **pick a point**

Prerequisite: A polyline

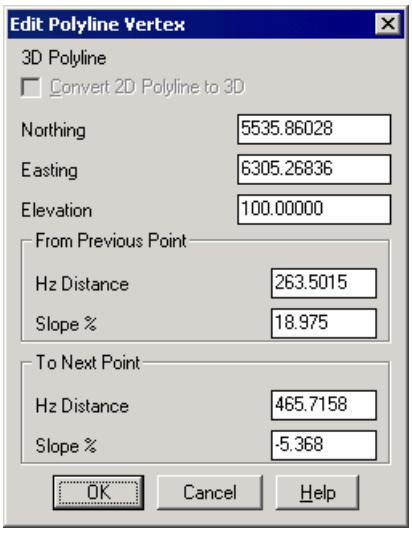
Keyboard Command: ADDPL

Edit Polyline Vertex

Function

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyine. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.



Prompts

1 Select polyline to edit: pick a 2D or 3D polyline

After you pick a polyline, the vertices are marked with a temporary X symbol

2 Pick point on polyline to edit: **pick the vertex to edit**

3 Dialog box is shown

4 Select polyline vertex to edit: press enter to quit or select another polyline

Prerequisite: A polyline

Keyboard Command: EDITPL

Edit Polyline Section

Function

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a

point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

Select contour to edit: *pick the contour polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

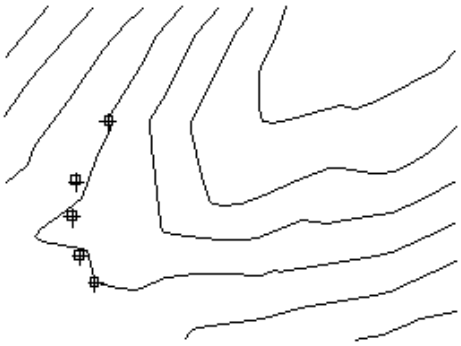
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *Press Enter*

Pick reconnection point on contour: *pick the contour polyline at the place to join*

Keyboard Command: editpl2

Prerequisite: polylines with elevation (contour polylines)





Edit this contour by picking new points
Contour with segment replaced with new points

Remove Duplicate Polylines

Function

This command finds any duplicate linework in your selection and removes all but one set.

Prerequisite: duplicate linework

Keyboard Command: rmduplwork

Remove Polyline Arcs

Function

This command allows you to replace arc segments in polylines with chords. Removing arcs is a prerequisite to some commands that don't handle arcs, such as Break by Closed Polyline and Make 3D Grid file. This command can add many vertices to the polyline.

Prompts

1 Select polylines to remove arcs from.

Select objects: **pick polylines**

2 Offset cutoff <0.5>: **Press Enter**

This specifies the maximum distance that any point on the arc will be allowed to shift.

Prerequisite: A polyline

Keyboard Command: RMARC

Remove Polyline Segment

Function

This command allows you to remove a specified segment from a polyline. (A polyline segment is the section between two vertices of the polyline.) There are two options for removing the segment. When you specify the Continuous option, the two vertices of the removed segments are averaged together to keep the polyline continuous. When you specify the Break option, the segment is left missing in the polyline, resulting in two separate polylines.

Prompts

1 Break polyline at removal or keep continuous [<Break>/Continuous]? **C**

2 Select polyline segment to remove: **pick point on polyline segment**

Prerequisite: A polyline

Keyboard Command: REMOVEPL

Remove Polyline Vertex

Function

This command allows you to remove the selected vertex from a polyline.

Prompts

1 Select polyline vertex to remove: **pick point on polyline**

Select the vertex to remove

Prerequisite: A polyline

Keyboard Command: RMVERTEX

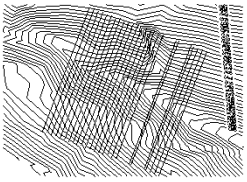
Break 3D Polyline by Surface

Function

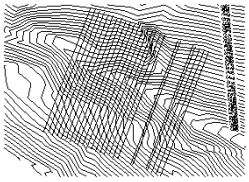
This command breaks 3D polylines at the intersection with a surface.

Prompts

Select Surface
Select polylines to clip.
Select objects: *pick the 3D polylines*
Erase polyline below surface [<Yes>/No]? *press Enter* If you answer yes, the segments of the polylines below the surface will be erased from the intersection, if any, of the polyline with the surface. Otherwise the polylines will only be broken into separate polylines at the intersection.



After Break 3D Polyline by Surface



Before Break 3D Polyline by Surface

Keyboard Command: surfbreak
Prerequisite: 3D Polylines to break and a surface.

Tag Hard Breakline Polylines

Function

This command tags polylines with a description so that Triangulate & Contour can identify these polylines as hard breaklines. The tag is invisible and doesn't change the polyline. Triangulate & Contour will not smooth the contours as they crosses these hard breaklines. For example you could tag 3D polylines that represent a wall so that the contours go straight across the wall without smoothing curves.

Prompts

Select hard breaklines.

Select objects: *select polylines*

Prerequisite: Polylines.

Keyboard Command: hardbrk

Untag Hard Breakline Polylines

Function

This command removes description tags from polylines. These tags are used by Triangulate & Contour to identify polylines as hard breaklines. Contours are not smoothed as they cross these hard breaklines. This routine untags polylines so that contours are smoothed across them.

Prompts

Select polylines to remove hard breakline tag from.

Select objects: *select polylines*

Prerequisite: Polylines with hard breakline tag.

Keyboard Command: softbrk

Smooth Polyline

Function

This command allows you to smooth selected polylines using a modified Bezier method. The resulting polyline passes through all the original points, and only the segments between the original points are smoothed. The looping factor controls the smoothing amount. A higher factor gives more looping.

Prompts

1 Enter the looping factor (1-10) <5>: **7**

2 Enter the offset cutoff <0.05>: **Press Enter**

3 Select polylines to smooth.

Select objects: **pick polylines**

Prerequisite: A polyline

Keyboard Command: SMOOTHPL

Change Polyline Width

Function

This command sets the width of the selected polylines. You can change the width of multiple polylines at one time.

Prompts

1 New width <1.0>: **2**

2 Select Contours/Plines to change width of:

Select objects: **pick polylines**

Prerequisite: A polyline

Keyboard Command: CWIDTH

Check Elevation Range

Function

This command finds elevations outside a specified range and highlights them on the drawing for easy viewing. This is useful for finding polylines with incorrect elevations.

You will be prompted for the minimum and maximum elevations in the range and than for the polyline(s) to check. You can either pick on individual polylines, window around a group polylines, or type in ALL for all the polylines, when you are done press ENTER. Next you can set the polylines outside the range to zero elevation so that they are easier to edit, or you say No to keep there elevations. When you are done than every elevation outside the range will be highlighted, if you chance the zoom of the drawing the blips will disappear.

Prompts

Command: checkpl

Enter elevation range minimum: 800

Enter elevation range maximum: 850

Select polylines to check.
Select objects: Specify opposite corner: 3310 found
3179 were filtered out.

Select objects: 1 found, 131 total
1 was filtered out.

Select objects:

Found 8 polylines outside of elevation range.
Set polylines outside elevation range to zero elevation [Yes/<No>]?

Prerequisite: elevations

Keyboard Command: checkpl

Close Polylines

Function

This command allows you to close a selection set of open polylines.

Prerequisite: Open polyline(s).

Keyboard Command: CLOSEPL

Open Polylines

Function

This command allows you to open a selection set of closed polylines.

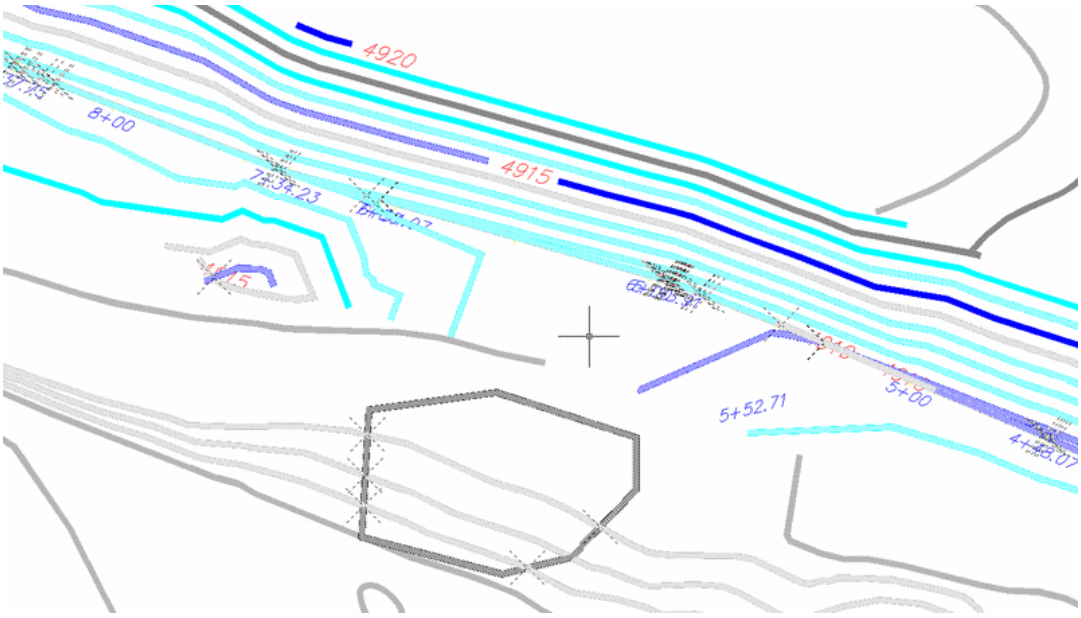
Prerequisite: Closed polyline(s).

Keyboard Command: OPENPL

Highlight Crossing Plines

Function

This command highlights selected polylines that are crossing in the drawing. Every intersection point where the polylines cross are marked with an X. A report is provided at the end where the X and Y of the intersection points are displayed with the two Z values and the Z difference.



Carlson Software Edit : c:\Carlson2007\USER\scadrprt.tmp

File Edit Settings

Open Save Print Exit Find Screen Hide

Crossing Breaklines Report 4/4/2006 15:00

Intersection	X	Y	Z1	Z2	Delta Z
1	544093.6085	160501.4266	1540.0000	0.0000	1540.0000
2	544244.8235	160540.6909	1540.0000	0.0000	1540.0000
3	544244.7077	160538.8089	1540.0000	0.0000	1540.0000
4	544158.7792	160696.2540	1540.0000	0.0000	1540.0000
5	544249.8746	160816.5767	1520.0000	0.0000	1520.0000
6	544332.9374	160495.1421	1520.0000	0.0000	1520.0000
7	543998.8958	160456.7193	1520.0000	0.0000	1520.0000
8	544154.6199	160751.1884	1520.0000	0.0000	1520.0000
9	544191.4615	160816.5767	1520.0000	0.0000	1520.0000
10	544990.2285	160281.4517	1500.0000	0.0000	1500.0000
11	544146.4027	160797.0493	1500.0000	0.0000	1500.0000
12	543927.0323	160425.3034	1500.0000	0.0000	1500.0000
13	544426.2473	160413.1184	1500.0000	0.0000	1500.0000

Prompts

Select polylines to check.
Select objects: Specify opposite corner: 32 found
Reading points ... 1677
Finding points on breaklines ...
19 crossing polylines are highlighted.
Minimum delta Z to report <0.0>: 2
Add polyline vertices at intersections with average elevation [Yes/<No>]? *N*

Keyboard Command: xing_plines
Prerequisite: Polylines with elevations

Select by Filter

Function

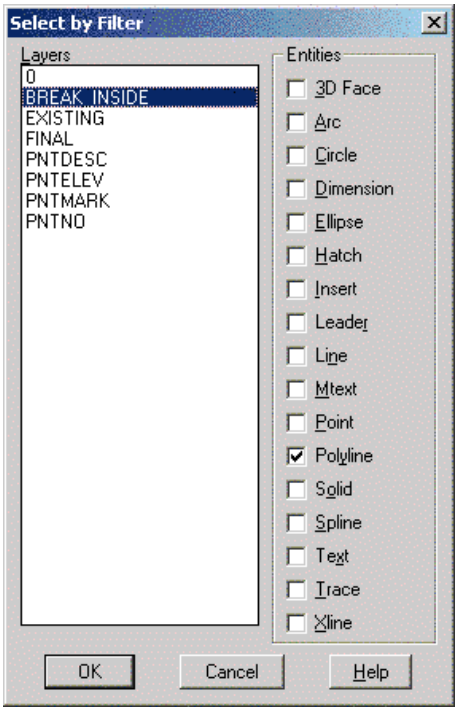
This tool can be used to build a selection set of objects inside a drawing based on layer and entity type. When the tool is executed it displays the following dialog.

Select the layer(s) on the left you wish to select, then turn on the toggle(s) for the entity types to consider. The tool then builds a selection set of those objects that resides on those layers.

When you execute your command following this selection building process, when you are prompted to select objects simply enter "P" for previous.

Prerequisite: none

Keyboard Command: fsel



Select by Elevation

Function

This command builds a selection set using elevations. You can select entities by all entities greater or less than the given number are selected and entities within the exclusion polylines are not selected.

Prompts

Select by greater, less or between elevations [<Greater>/Less/<Between>]? *g*

Enter elevation for greater than: *99*

Ignore zero elevations [<Yes>/No]? *Y*

Select objects to build selection set.
Select objects: *all*
76 found

Select objects: *Press Enter*

Processing selection set ...
Built selection of 25 objects for elev more than 99.00.
To use type 'P' at Select objects: prompt.

Command: *Erase*

Select objects: *P* To use previous selection set created by Select by Area

43 found

Select objects: *Press Enter*

Prerequisite: entities with elevation

Keyboard Command: ZSELECT

Select by Area

Function

This command builds a selection set using inclusion and/or exclusion closed polylines. Entities within the inclusion polylines are selected and entities within the exclusion polylines are not selected. With the Window selection method, the entity must be entirely inside the inclusion area and entirely outside the exclusion area to be included in the selection set. With the Crossing selection method, an entity is added to the selection set if any part of the entity is inside the inclusion area.

Prompts

Select the Inclusion perimeter polylines (Enter for None).

Select objects: *pick the closed polyline*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines (Enter for None).

Select objects: *Press Enter*

Type of selection (Window/<Crossing>)? *Press Enter* for crossing

Select objects to build selection set.
Select objects: *All* These selected objects are checked with the inclusion/exclusion polylines.

Select objects: *Press Enter*

Built selection set with 43 objects.

Command: *Erase*

Select objects: *P* To use previous selection set created by Select by Area

43 found

Select objects: *Press Enter*

Prerequisite: closed perimeter polylines

Keyboard Command: ssgetarea

View Menu 6

Redraw

Function

This command refreshes the display in the current viewport.

Prerequisite: None

Keyboard Command: R

Regenerate

Function

This command regenerates the drawing and refreshes the current viewport.

Prerequisite: None

Keyboard Command: REGEN

Zoom - Window

Function

This commands zooms to display an area you specify by two opposite corners of a rectangular window.

Prerequisite: None

Keyboard Command: ZOOM, W

Zoom - Dynamic

Function

This command zooms to display the generated portion of the drawing using a view box. The view box represents your viewport, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms the viewport, filling it with the image inside the view box.

Prerequisite: None

Keyboard Command: ZOOM, D

Zoom - Previous

Function

This command zooms to display a previous view. You can restore up to 10 previous views.

Prerequisite: None

Keyboard Command: ZOOM, P

Zoom - Center

Function

This command zooms to display a window you define by picking a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the magnification.

Prompts

1 Specify center point: **pick a point**

2 Enter magnification or height <226.66>: **enter a value**

Prerequisite: None

Keyboard Command: ZOOM, C

Zoom - Extents

Function

This command zooms to display the drawing extents. You can use Zoom Extents transparently, but it always regenerates the drawing.

Prerequisite: None

Keyboard Command: ZOOM, E

Zoom IN

Function

This command increases the zoom factor of the current viewport by a factor of 2.0.

Prerequisite: None

Keyboard Command: ZOOM, 2.0x

Zoom OUT

Function

This command decreases the zoom factor of the current viewport by a factor of 0.5.

Prerequisite: None

Keyboard Command: ZOOM, 0.5x

Zoom Point(s)

Function

This command centers the screen around a point, or range of points, you specify using either the point number or description. The command searches the current coordinate file. In addition to centering the screen, you can also change the magnification value. The default value is the current magnification. To zoom in, enter a smaller value. To zoom out, enter a larger value.

Prompts

1 Find by point number or description [<Number>/Desc]: **Press Enter**

2 Point number or range of point numbers to find <1>: **8-12**

3 Magnification or Height <323.00>: **Press Enter**

Prerequisite: Current coordinate (.CRD) file

Keyboard Command: ZOOMPNT

Pan

Function

This command moves the drawing display in the current viewport. The cursor changes to a hand cursor. By holding down the pick button on the pointing device, you lock the cursor to its current location relative to the viewport coordinate system. The drawing display is moved in the same direction as the cursor.


hand cursor

When you reach a logical extent (the edge of the drawing space), a bar is displayed on the hand cursor on the side where the extent has been reached. Depending on whether the logical extent is at the top, bottom, or side of the drawing, the bar is either horizontal (top or bottom) or vertical (left or right side).


top extent


right extent


bottom extent


left extent

When you release the pick button, panning stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again to pan the display from that location.

To stop panning at any time, press Enter or ESC.

Prerequisite: None

Keyboard Command: P

Twist Screen Standard

Function

This command allows you to "twist" the screen's orientation so that a direction other than North is toward the top of the screen and the drawing. It does not do a coordinate rotation, and it leaves the database unchanged. The ROTATE and MOVE commands in the Edit menu can be used to do a coordinate rotation and translation.

This command prompts you for the twist angle, then adjusts the screen and crosshairs to that angle. The twist angle is always measured counterclockwise, with 0 degrees at the east/right.

Prerequisite: None

Keyboard Command: TWIST1

Twist Screen Line

Function

This command is a variation of Twist Screen Standard. The command aligns a selected line, polyline or text to be parallel to the east-west direction of your graphics screen.

Think of what you select as a pointer or arrow that will be moved to point in the east direction of the screen. Select the line, polyline or text closest to the endpoint that you want it to point in the horizontal or east direction of the screen.

Prerequisite: None

Keyboard Command: TWIST2

Twist Screen Surveyor

Function

This command is another variation of Twist Screen Standard. You enter the angle/azimuth that you want to be aligned parallel to the east-west direction of the graphics screen.

Prerequisite: None

Keyboard Command: TWIST3

Restore Due North

Function

This command twists the screen to make due north vertical.

Prerequisite: None

Keyboard Command: TWIST4

Display Order

Function

This command allows you to change the display order of objects by repositioning an entity from either the background to the forefront of the drawing view or from the forefront to the background of the drawing view.

Prerequisite: None

Keyboard Command: draworder

Update Colors For Set Elevations

Function

This command refreshes the color of entities depending on their elevation and layer target. For entities assigned to the Existing or Design layer targets, if the entities are at zero elevation then their color is set to grey. Otherwise the entities have their true, original color. If the Automatic Update Colors command under Settings->Configure->Takeoff Module is toggled off, then this command is the way to update the entity colors after editing elevations.

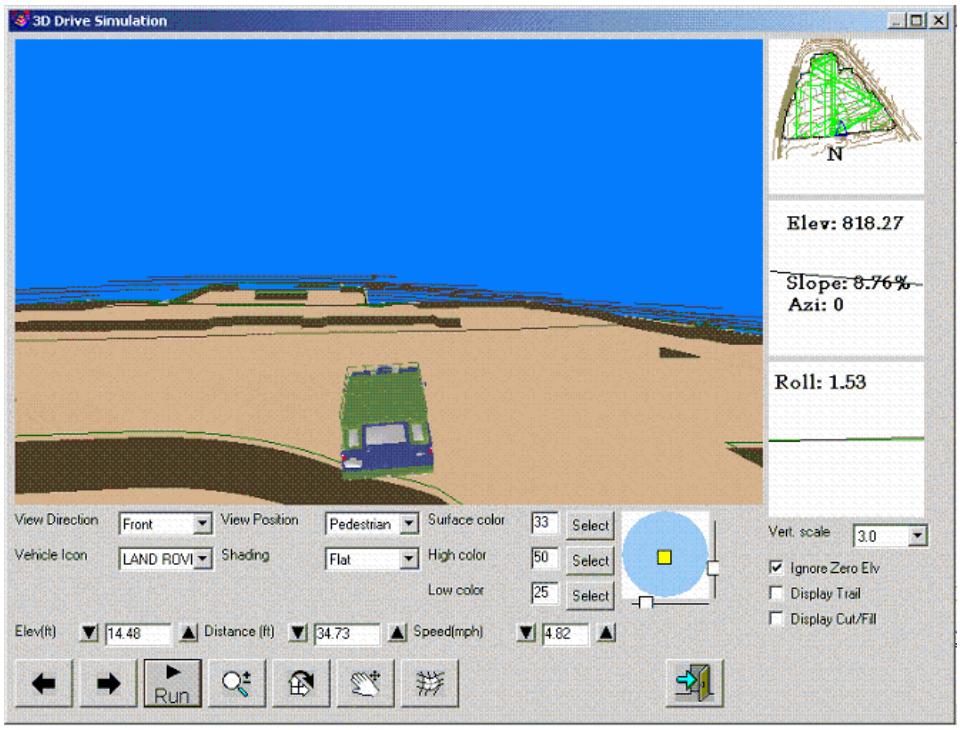
Prerequisites: none

Keyboard Command: update_tk_colors

3D Drive Simutation

Function

This command allows you to view and move around the design surface in 3D mode.



Use the arrows on your keypad to move around the drawing.

At the very bottom of the window you will find the basic commands: Run will start to drive your vehicle around the surface, once your vehicle is moving the Run button turns into the Stop button. The arrows moves your vehicle left and right. The magnify glass zooms in and out. Click and drag up to zoom in and click and drag down to zoom out. When your vehicle is stopped the  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis.

The hand icon allows you to pan around the viewer. Click and drag the direction you want to move. The  icon toggles the shading of the surfaces. The  icon exits 3D Driver Simulation.
Above the basic command buttons you can change the Elevation and Distance away from your vehicle. Also, you can set the speed at which your vehicle travels. For a smaller drawing you may want to move around slower, for a larger drawing faster. Note: Unrealistic speeds such as 500 mph in a doozer may cause 3D Drive Simulation to freeze.

View Direction: Sets the direction of the view from the Front, Back, Left, or Right.
Vehicle Icon: You can select which Vehicle you want to use whether: Doozer, Hummer, School Bus or none at all.
View Position: Sets the elevation and distance to either that of the driver, a pedestrian, or bird.
Shading: Here you can set the shading of the surface to either Flat, Smooth, Elevation, Cut/Fill, or none. Flat just shades the contours as the are. Smooth smooths contours to look for realistic. Elevation colors different elevations in different colors so differences can visual be seen. Cut/Fill colors areas of cut differently than areas of fill so they can be visually seen. None merely shows the triangulation.and does not shade in a surface.
You can select the Surface, High, and Low color by enter in an AutoCAD defined color number or you can choose Select to pick a color. The circle on the right determines the shade of the color.

In the top right of the 3D viewer is an aerial map of your surface. Below that the Elevation, Slope percentage, Azimuth, and Roll are updated as your vehicle moves around the surface. Slope and Roll are shown visually here as well.

On the bottom right you can set the Vertical Scale and check to Ignore Zero Elev, Display Trail, and Display Cut/Fill. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Display Trail draws a line where your vehicle has driven. Display Cut/Fill displays the cut and the fill.

Prompts

Command: tk_flyby

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles
Loading reference file...

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading entities...
Loading vehicle icon file...

Loading edges...
Loaded 926 points and 2150 edges

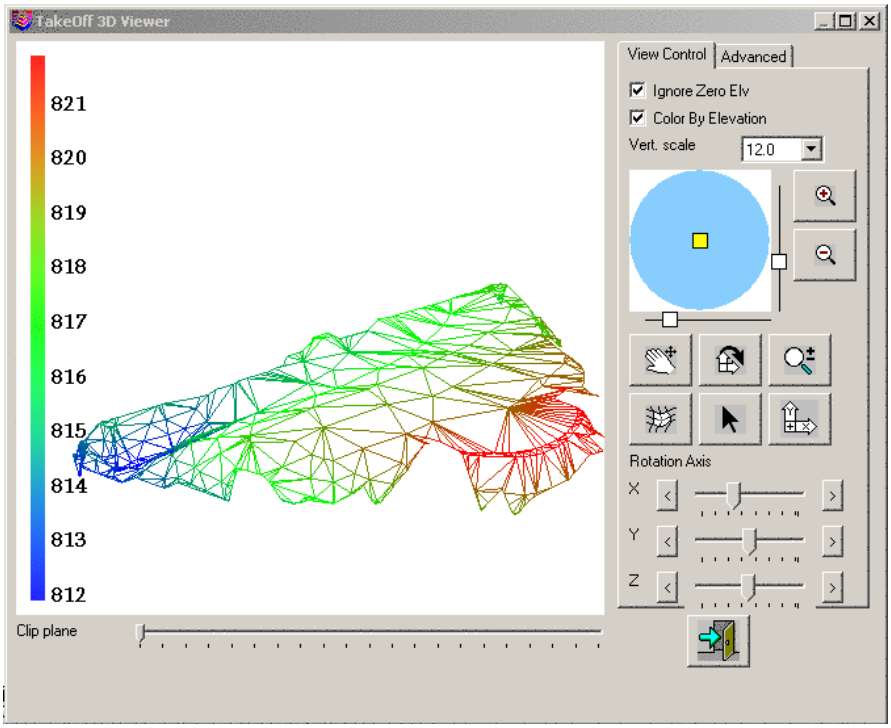
Prerequisite: a design surface

Keyboard Command: tk_flyby

Existing Surface 3D Viewer

Function

This command allows you to view the existing surface in 3D mode.



In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not displayelevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on theplus magnifyglass to zoom in and

theminus magnifyglass to zoomout. With the  icon click and drag up to zoom in and drag down to zoomout. The hand icon below the color circle allows you to pan around theviewer. Click and drag the direction you want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you movethe cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the zaxis.The  icon toggles on and off the shading of the surface. The arrow icon reports the elevations at the bottom of the screenas you move around the surface. The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prompts

Command: cube_exist

Loading entities...
Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

Loading edges...
Loaded 574 points and 1393 edges

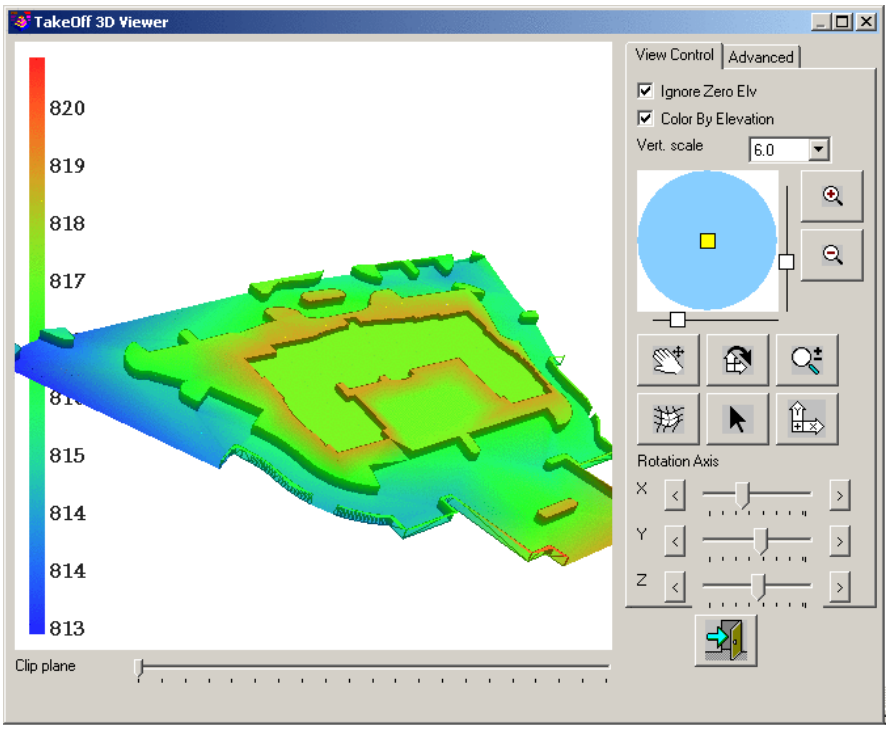
Prerequisite: a existing surface

Keyboard Command: cube_exist

Design Surface 3D Viewer

Function

This command allows you to view the design surface in 3D mode.



In the top right of the control bar you can check to Ignore Zero Elev and Color By Elevation and change the Vertical Scale. If you increase the Vertical Scale than elevation differences can be seen easier. Ignore Zero Elev does not display elevations of zero in the 3D viewer. Color By Elevation shows elevation change with the change of colors. Note: Color By Elevation is used in the above example. To adjust the color use the color circle on the right.

The magnify glass icons can be used to zoom in and out. Click on the plus magnify glass to zoom in and the minus magnify glass to zoom out. With the  icon click and drag up to zoom in and drag down to zoom out. The hand icon below the color circle allows you to pan around the viewer. Click and drag the direction you want to move. The  icon can be used to rotate the vantage point of the viewer by the x, y, or z axis. When you move the cursor to the screen it will change into a x, y symbol or a z symbol. Move the cursor around to move it from one to the other. If you have the x, y cursor move right or left to change the x axis view, or to change the y move the cursor up or down. If you have the z cursor than move it in a circular fashion to rotate the view point according to the z axis. The  icon toggles on and off the shading of the surface (the shading is shown in the above drawing). The arrow icon reports the elevations at the bottom of the screen as you move around the surface. The  icon restores the surface viewpoint to flat. The  icon exits 3D Driver Simulation.

Rotation Axis: These three control bars rotate the surface around the x, y, and z axis. Clip plane trims the size of the surface shown in the viewer.

Prompts

Command: cube_design

Loading entities...
Loading edges...
Loaded 5057 points and 14923 edges

Prerequisite: a design surface

Keyboard Command: cube_design

FlyOver Along 3D Polyline

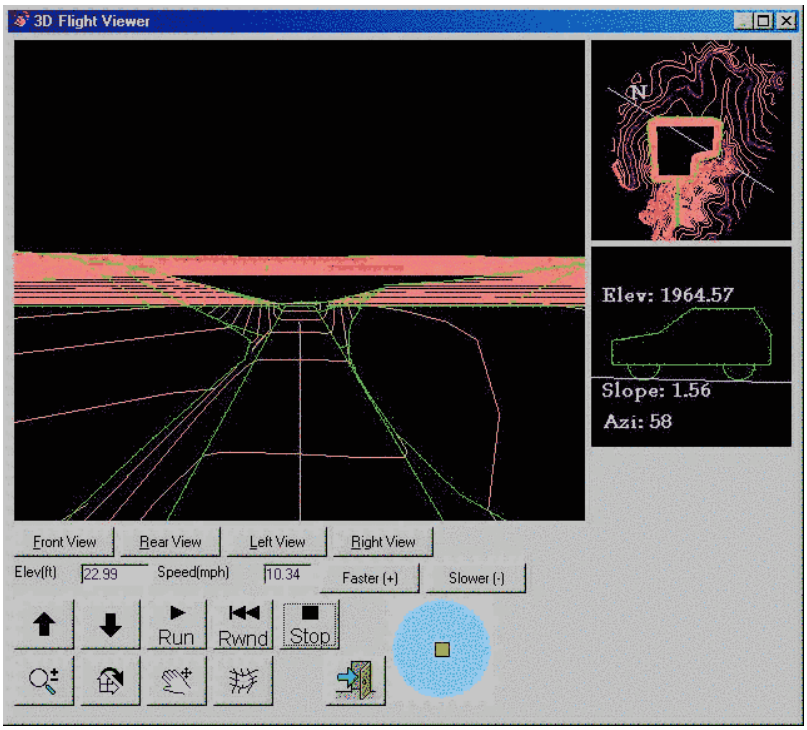
Function

This command allows you to view a self guided animation of following a path through a 3D surface model. There are two variations to this command. When the command is started, you must specify whether you want to use a surface model from file or screen entities.

Surface model from file: Using this method, you can select either a triangulation (.TIN) file or a grid (.GRD) file, then you have the option of following a polyline or following a "free" path. If you choose the polyline method, then the animation is limited to following the polyline. If you choose the "free" path method, you first specify two points to obtain a starting direction, the while inside the viewer you can point the animation in any direction.

Screen entities: Using this method, you must select a 3D polyline to follow. The animation is limited to following the polyline.

After making the above selections, the 3D graphics window is opened. The main window is for the animation, the smaller upper right window shows you the overall plan view, and the smaller window located at middle right shows you the current elevation, slope and azimuth. While following a "free" path, you will have a 3rd small window located at lower right which shows you the amount of roll at your current position.





This button raises the elevation of your viewing position.



This button lowers the elevation of your viewing position.



This button turns your viewing position to the left.



This button turns your viewing position to the right.



This button allows you to zoom in and out.



This button allows you to rotate the main animation window in any X, Y or Z direction.



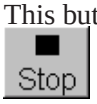
This button allows you to pan.



This button toggles shading on and off.



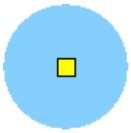
This button starts the animation in the main window.



This button stops the animation.



This button exits the 3D Surface FlyOver command



Control for position of the light source, viewed from above.

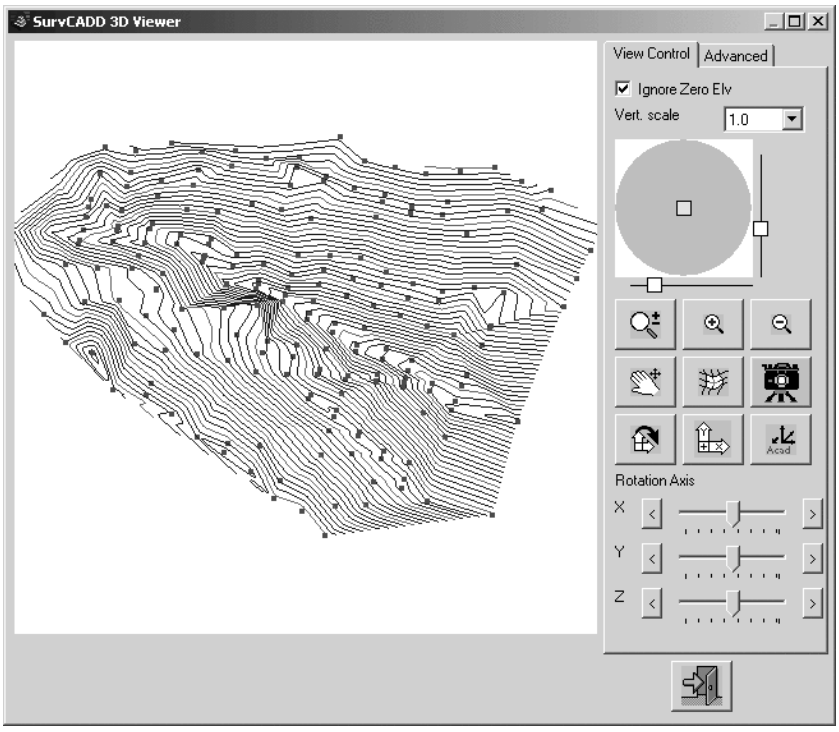
Prerequisite: Surface Model and optionally a 3D Polyline

Keyboard Command: flyby

3D Viewer Window

Function

This command allows you to view the selected 3D faces, polylines, lines, and points in 3D using the OpenGL graphics library for rendering, which gives it superior performance. This command gives you the ability to zoom in and out, pan, rotate around the X, Y, or Z axis, and shade in user-positioned lighting. You also have the option to export the view to a windows bitmap (.BMP) file. The window below is displayed after you select the entities to be shown. To pan the display, click on the image and drag the pointer.



- **Controls**: Light intensity for the direction and ambient light.
- **Block Models**: Rendering method options were added along with the ability to turn off entities by color.
- **Ignore Zero Elv**: This option excludes entities on zero elevation from the display.
- **Vert**. **Scale**: Controls the vertical exaggeration of the 3D display.
- **Light Position**: This control represents the position of the sun in the sky looking from above. The position of the sun in the center of the control means the sun is in a zenith, and a position near the edge of the circle means the sun is near the horizon. To move the sun, simply drag it to new location or click there.
- **Dynamic Zoom Mode**: Allows you to zoom in and out by holding the left mouse button down and moving the mouse up and down.
- **Zoom In**: This option incrementally zooms in to the view.
- **Zoom Out**: This option incrementally zooms out to the view.
- **Pan Mode**: Allows you to pan the display
- **Shade Model**: This option allows you to shade the surfaces of your view. The color of the shade depends on the lighting of the surface, which you adjust using the Light Position control in the middle of the controls window. See Light Position below.
- **Export to BMP**: This option creates a .BMP bitmap file from your current view.
- **Rotation Mode**: Allows you to rotate the view in the X or Y axis while the cursor is in the middle of the display. Rotates on the Z axis when the cursor is near any edge of the display. The cursor will indicate the axis.
- **Plan View**: This option restores the X, Y, and Z axes back to zero.
- **Set Acad View**: This option allows you to send the current 3D view back to the TakeOff screen when you exit the 3D Viewer window. You update the current CAD view to match the view that is displayed in the 3D viewer window.

- **Z-Axis**: This slider rotates the view on the Z axis. The range of these sliders is -180 to +180 degrees with middle being 0, which is the default position when the viewer starts.
- **Y-Axis**: This slider rotates the view on the Y axis.
- **X-Axis**: This slider rotates the view on the X axis.
- **General**: This tool uses the same background color as the AutoCAD graphics window.

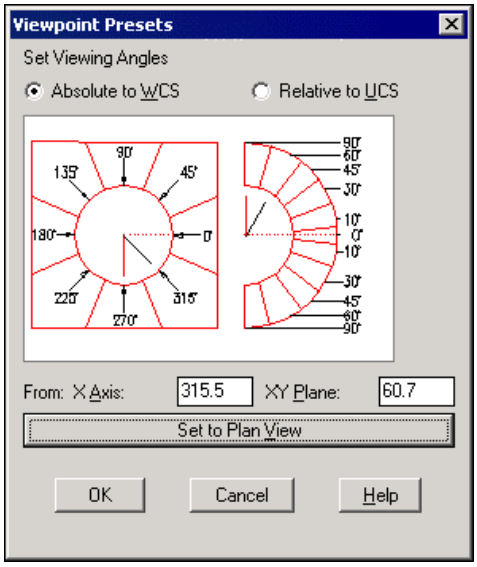
Prerequisite: Entities to display.

Keyboard Command: CUBE

Viewpoint 3D

Function

This command allows you to define 3D view settings.



1 Under Set Viewing Angles, you must set the direction of the view relative to either the world coordinate system (WCS) or a user coordinate system (UCS).

- **Absolute to WCS**: This option sets the view direction relative to the WCS.
- **Relative to UCS**: This option sets the view direction relative to the current UCS.

2 You must specify the viewing angles.

- **X Axis**: This field specifies the angle from the X axis.
- **XY Plane**: This field specifies the angle from the XY plane. You can also use the sample image to specify viewing angles. The black arm indicates the new angle. The red arm indicates the current angle. Specify an angle by selecting the inner region of the circle or half-circle. Selecting the bounded outer regions rounds off the angle to the value displayed in that region.
- **Set to Plan View**: This option sets the viewing angles to display the plan view relative to the selected coordinate system.

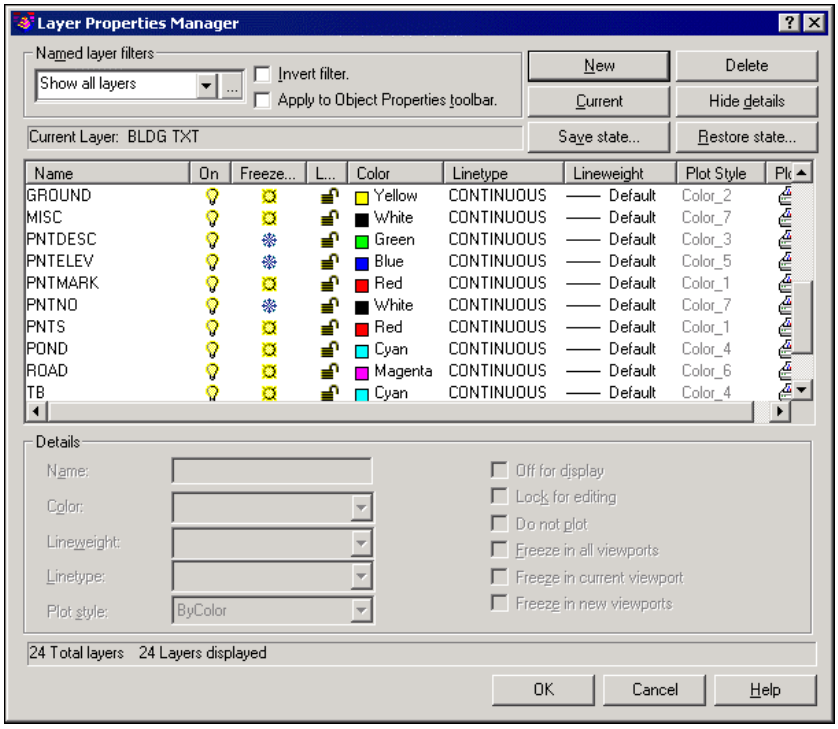
Prerequisite: None.

Keyboard Command: DDVPOINT

Layer Control

Function

This command allows you to manage layers and layer properties.



This Layer Properties Manager dialog box makes a layer current, adds new layers to the layer name list, and renames an existing layer. You can assign properties to layers, turn layers on and off, freeze and thaw layers globally or by viewport, lock and unlock layers, set plot styles for layers, and turn plotting on and off for layers. You can filter the layer names displayed in the Layer Properties Manager, and you can save and restore layer states and properties settings.

1 Under Named Layer Filters, you determine which layers to display in the list of layers. You can filter layers based on whether they're xref-dependent, or whether they contain objects. You can also filter layers based on name, visibility, color, linetype, lineweight, plot style name, whether they are plotted, or whether they are frozen in the current viewport or in new viewports.

- **[...]**: This button displays the Named Layer Filters dialog box.
- **Invert Filter**: This option displays layers based on the opposites of the criteria you select when you are using a named layer filter. Layers that fit the inverse criteria are displayed in the layer name list.
- **Apply to Object Properties Toolbar**: This option displays in the Object Properties toolbar only layers that match the current filter. The layer list tooltip on the Object Properties toolbar displays the filter status of layers in the drawing. (To display the layer list tooltip, position the pointing device over the layer list on the Object Properties toolbar.)
- **New**: This option creates a new layer. After you choose New, the list displays a layer named LAYER1. You can edit this layer immediately. To create multiple layers quickly, you can select a layer name for editing and enter

multiple layer names separated by commas. If you create a new layer, the new layer inherits the properties of the currently selected layer in the layer list (such as Color, and On/Off state). To create layers with default settings, make sure that there are no selected layers in the list or that you select a layer with default settings before beginning layer creation.

- **Current**: This option sets the selected layer as the current layer. The CLAYER system variable stores the layer name.

- **Delete**: This option deletes selected layers from the drawing file definition. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects (including objects in block definitions), the current layer, and xref-dependent layers. Layers that don't contain objects (including objects in block definitions), are not current, and are not xref-dependent can be deleted by using the PURGE command. Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

- **Show/Hide Details**: This option controls whether the Details section is displayed in the Layer Properties Manager.

- **Save State**: This option displays the Save Layer States dialog box, in which you save layer state and layer properties settings of all layers in a drawing. You can choose which layer states and properties you want to preserve. You save a layer state by assigning it a name.

- **Restore State**: This option displays the Layer States Manager, in which you can manage named layer states.

2 The Layer Properties Manager dialog box displays all layers and their properties. To modify a property, click its icon. To quickly select all layers, right-click your pointing device and use the shortcut menu. The following are the layer properties you can modify:

- **Name**: This field displays the names of the layers. You can select a name, and then click and enter a new name.

- **On/Off**: This field turns layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

- **Freeze/Thaw in All Viewports**: This field freezes and thaws layers in all floating viewports. A frozen layer is invisible and excluded from regeneration, hiding objects, rendering, and plotting. A thawed layer is visible and available for regeneration, hiding objects, rendering, and plotting.

You can freeze layers to speed up ZOOM, PAN, and many other operations, improve object selection performance, and reduce regeneration time for complex drawings. TakeOff does not display, plot, or regenerate objects on frozen layers. Objects on frozen layers do not hide objects and are not rendered.

You can freeze layers in all viewports, in the current viewport, or in new viewports.

Freeze layers that you want to be invisible for long periods. When you thaw a frozen layer, the program regenerates and displays the objects on that layer. If you switch between visible and invisible states frequently, use the On/Off setting.

- **Lock/Unlock**: This field locks and unlocks the layers. You cannot select or edit objects on a locked layer. Locking a layer is useful if you want to view information on a layer for reference but do not want to edit objects on that layer.

- **Color**: This field changes the color associated with the selected layers. Clicking the color name displays the Select Color dialog box.

- **Linetype**: This field changes the linetype associated with the selected layers. Clicking any linetype name displays the Select Linetype dialog box.

- **Lineweight**: This field changes the lineweight associated with the selected layers. Clicking any lineweight name displays the Lineweight dialog box.

- **Plot Style**: This field changes the plot style associated with the selected layers. If you are working with color-dependent plot styles (the PSTYLEPOLICY system variable is set to 1), you cannot change the plot style associated with a layer. Clicking any plot style displays the Select Plot Style dialog box.
- **Plot/Don't Plot**: This field controls whether the selected layers are plotted. If you turn off plotting for a layer, the objects on that layer are still displayed. Turning off plotting for a layer affects only visible layers in the drawing (layers that are on and thawed). If a layer is set to plot, but is currently frozen or off in the drawing, TakeOff does not plot the layer. Turning off plotting for layers containing reference information such as construction lines can be useful.

Prerequisite: None

Keyboard Command: LAYER

Change Layer

Function

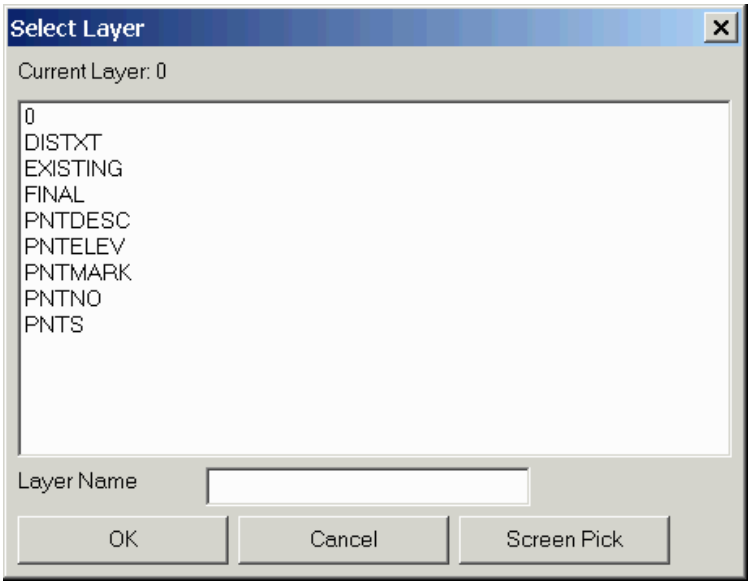
This command allows you to change the layer of a group of entities you select. The layer name can be either typed, selected from a list or read from an existing entity you select from the drawing.

Prompts

Select entities to be changed.

Select objects: **pick entities**

Select Layer Dialog



Prerequisite: None

Keyboard Command: lchg

Freeze Layer

Function

This command allows you to freeze layers assigned to the entities you select.

Prerequisite: None

Keyboard Command: LOFF

Thaw Layer

Function

This command allows you to thaw the layers frozen by the Freeze Layer command.

Prerequisite: None

Keyboard Command: LON

Isolate Layer

Function

This command allows you to freeze all layers except the layers assigned to the entities you select.

Prompts

Select objects on layers to isolate.

Select objects: **pick entities**

Prerequisite: None

Keyboard Command: ISOLATE

Restore Layer

Function

This command thaws the layers frozen by the Isolate Layer command.

Prerequisite: You must have previously run the Isolate Layer command

Keyboard Command: RESTORE

Set Layer

Function

This command allows you to change the current layer by selecting an entity on the desired layer.

Prerequisite: Entity to select.

Keyboard Command: LSET

Draw Menu 7

Line

Function

This command allows you to draw a line entity by picking points on the screen or by supplying the coordinate values using the point number and associated coordinates stored in the current coordinate file. The Line command links the line with the points when the line is drawn using point numbers if the Link Linework with Points option is turned on. This option is set under General Settings in the Configure command in the Settings menu. With links active, changing a point with a command like Move Points automatically updates the line. This command always draws 2D lines with a zero elevation.

Prompts

1 Pick point or point numbers: 1-3

You may enter a single point number or a range of point numbers

2 Undo/Distance/<Pick point or point numbers>: 16

3 Undo/+/-/Close/Distance/<Pick point or point numbers>: 35

4 Undo/+/-/Close/Distance/<Pick point or point numbers>: +

The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

5 Perpendicular Distance Right: 80

6 Undo/+/-/Close/Distance/<Pick point or point numbers>: -

The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

7 Perpendicular Distance Left: 105.12

8 Undo/+/-/Close/Distance/<Pick point or point numbers>: D

The distance option allows you to input a distance for the next line segment. The position of the cursor determines the angle.

9 Enter distance: 174.32

10Undo/+/-/Close/Distance/<Pick point or point numbers>: C

The close option draws a line segment back to the original starting point

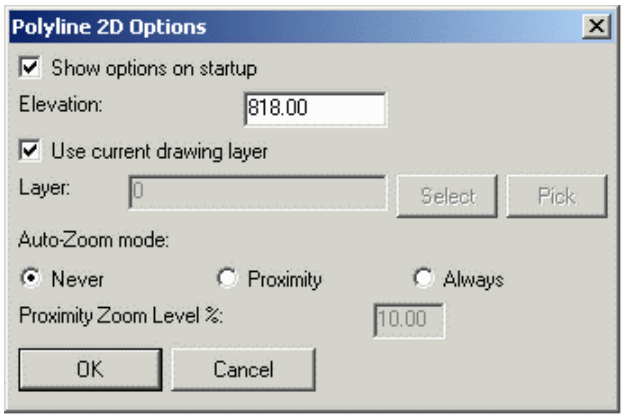
Prerequisite: None

Keyboard Command: 2DLINE

2D Polyline

Function

This command allows you to draw a polyline entity in a variety of ways, including: Continue, Extend, Follow, Arc, Direction, Close, Pick point, or point numbers. A Polyline is a series of line and/or arc segments joined together in one entity or object.



This dialog automatically comes up when you run the command. If you do not want it to come up ever time you run the command then toggle off the first dialog box called *Show options on startup*. In this dialog you can set the elevation of the polyline as well as the layer it will appear on. *Auto-Zoom mode* regenerates the screen as you draw your polyline to give you the best view as youare working. *Always* means it will always regenrate the screen. *Proximity* regenrates the screen according to the *Proximity ZoomLevel %*. *Never* toggles off this feature.

Prompts

[Continue/Extend/Follow/Options/<Pick point or point numbers>]:

Continue allows you to run Arc, Direction, Close, Extend, or Follow for an existingpolyline.

Extend allows you to extend an existing polyline by picking or entering in adistance.

Follow allows you to match the path of your polyline with an existing polyline. You are prompted toselect the point in which your polyline first intersects with the existing polyline and the point in which it exits. If the existing polyline is a closed polyline than you will be prompted to approve the direction in which your polyline follows the existing closed polyline.

Options brings up the *Polyline 2D Options* dialog thatappears when you first run the command.

Pick point allows you topick a point on the screen to start your polyline.

Pointnumbers allows you to enter in coordinates to start your polyline.

Arc allows you to draw an arc by Arc length/Chord/Radius/Second point in your polyline. The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line. This is useful for plotting buildings.

Distance for an angle code, a backsight point, an angle (dd.mmss), and a distance to draw your polyline.

Close will connect the ends of your polyine making it a closed polyline.

Prerequisite: None

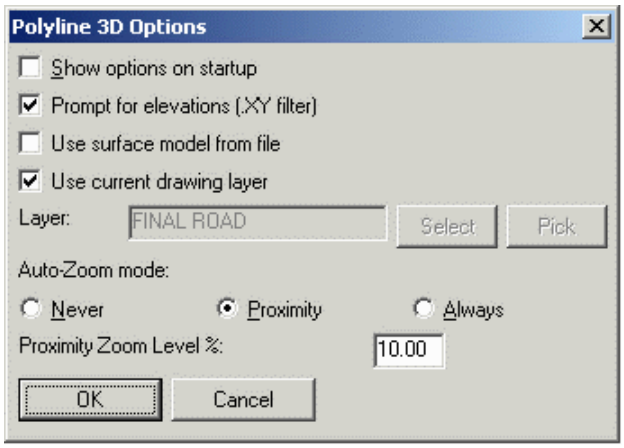
Keyboard Command: 2DP

3D Polyline

Function

This command allows you to set a specified layer and plot a 3D polyline which can be used to define breaklines and features to better define surface models. You can pick points from the graphics screen or use point numbers from the current coordinate file. You can also specify a range of point numbers. For example, entering "3-1,5,10" would draw 3D Line segments from point number 3 to 2 to 1 to 5 to 10.

You can choose to pick points from a surface model that is stored in a file, which can be either a grid (.GRD) file or triangulation (.TIN) file. When a point is picked or a point number is entered, the program uses the northing and easting of this point and calculates the elevation from the surface model. You can also draw a 3D polyline entity by Continue, Extend, Follow, Arc, Direction, or Close in this command. A Polyline is a series of line and/or arc segments joined together in one entity or object.



Prompts

[Continue/Extend/Follow/Options/<Pick point or point numbers>]:

Continue allows you to run Arc, Direction, Close, Extend, or Follow for an existingpolyline.

Extend allows you to extend an existing polyline by picking or entering in adistance.

Follow allows you to match the path of your polyline with an existing polyline. You are prompted toselect the point in which your polyline first intersects with the existing polyline and the point in which it exits. If the existing polyline is a closed polyline than you will be prompted to approve the direction in which your polyline follows the existing closed polyline.

Options brings up the *Polyline 2D Options* dialog thatappears when you first run the command.

Pick point allows you topick a point on the screen to start your polyline.

Pointnumbers allows you to enter in coordinates to start your polyline.

Arc allows you to draw an arc by Arc length/Chord/Radius/Second point in your polyline. The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line. This is useful for plotting buildings.

Distance for an angle code, a backsight point, an angle (dd.mmss), and a distance to draw your polyline.

Close will connect the ends of your polyine making it a closed polyline.

Prerequisite: None

Keyboard Command: 3DP

Circle

Function

This command allows you to draw a circle.

Prompts

1 Pick center point or point number or [3P/2P/TTR]: pick point or specify option

- 3P: This option draws a circle based on three points on the circumference.
- 2P: This option draws a circle based on two endpoints of the diameter.
- TTR-Tangent, Tangent, Radius: This option draws a circle with a specified radius tangent to two objects.

2 Specify radius of circle or [Diameter]: enter a value

Sometimes more than one circle matches the criteria specified in the command. The circle whose tangent points are closest to the selected points is drawn.

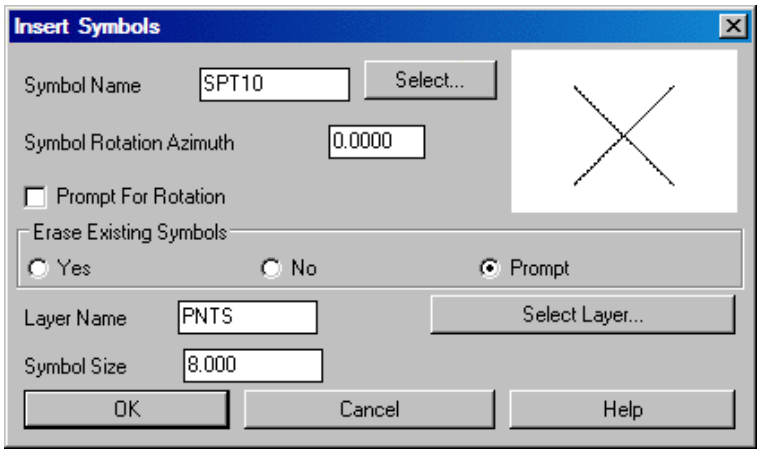
Prerequisite: None

Keyboard Command: SCIRCLE

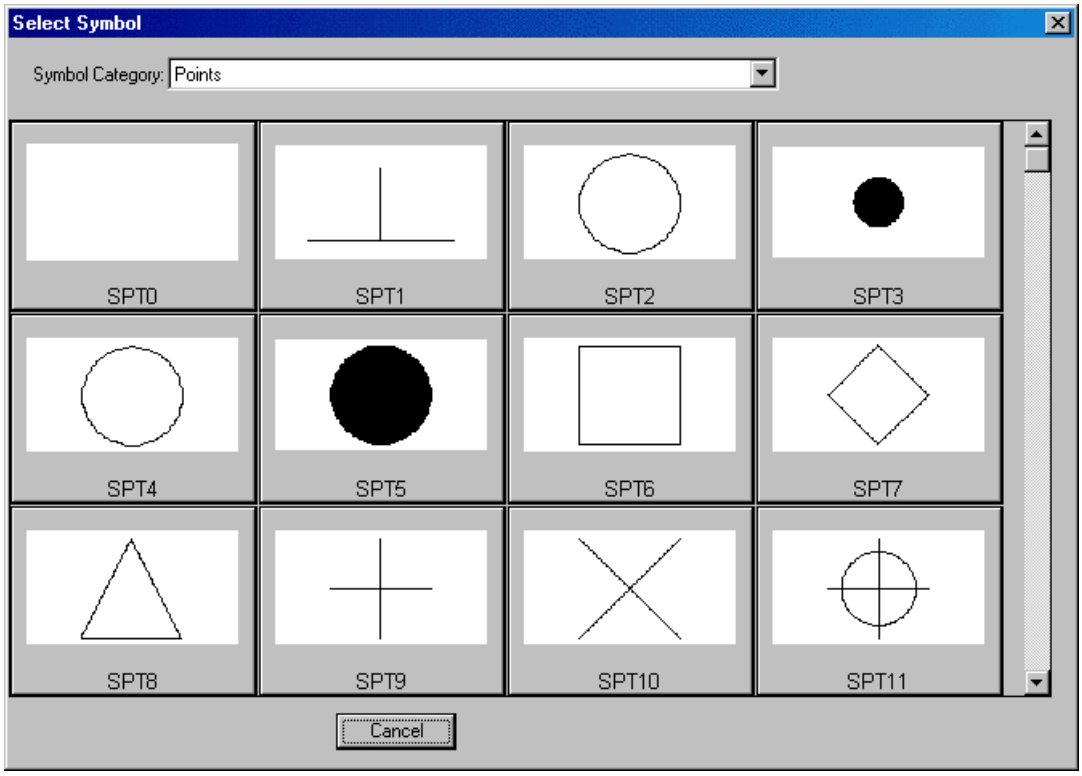
Insert Symbols

Function

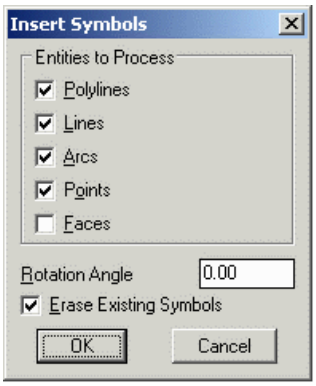
This command inserts symbols from the symbol library into the drawing. The symbol library may be edited using the *Edit Symbol Library* command. The locations for the symbols can be specified by picking points, specifying point numbers in the current coordinate (.CRD) file or by entering the northing and easting. If you specify a point number, and that point number already has a symbol on it, you will be prompted whether or not to replace the existing symbol. Selecting the Enter coords option allows you to insert the symbol by entering a northing, easting and elevation. Using the Select entities option, symbols can also be placed on arcs, faces, points, text, lines and polylines. Under the Options command, you can turn prompting for rotation on or off. With rotation off, the symbol will be inserted horizontal to the current twist screen. Choose a symbol from the Select Symbol dialog by clicking on it. The Symbol Category choices are Points, Trees and Map Symbols. You may select a category by choosing the Symbol Category dropdown list. Within each category, use the scroll bar to view all of the symbols.



Appears at start of command



Appears when Select (symbol) is chosen



Select entities dialog box

Prompts

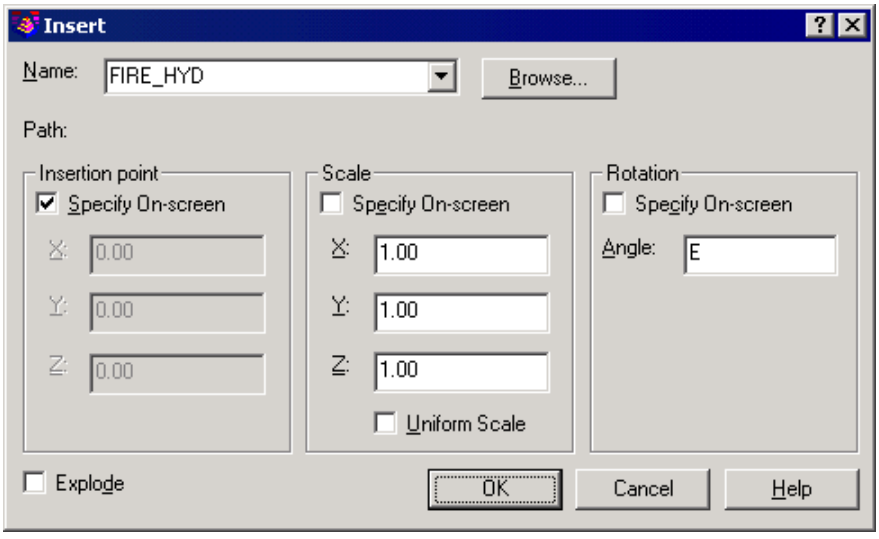
Insert Symbols dialog *choose variables and click OK*
Options/Select entities/Enter coords/<Pick point or point numbers>: *pick a point*
Options/Select entities/Enter coords/<Pick point or point numbers>: *5-10* Inserts symbols at points 5-10 from the current coordinate file.
Options/Select entities/Enter coords/<Pick point or point numbers>: *S*
Insert Symbols dialog
Select arcs, faces, points, text, lines and polylines. *select objects*
Options/Select entities/Enter coords/<Pick point or point numbers>: *press Enter to end*

Keyboard Command: ptsym
Prerequisite: None

Insert Drawing

Function

This command allows you to place a named block or drawing into the current drawing.



1 In the Insert dialog box, you specify the block to insert and define the position for the inserted block. The last block you insert during the current editing session becomes the default block for subsequent uses of this command.

- Name: This field specifies the name of a block to insert or the name of a file to insert as a block.
- Browse: This button opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or a file to insert.

2 Under Insertion Point, you specify the insertion point for the block.

- Specify On-Screen: This option specifies the insertion point of the block using the pointing device.
- X: This field sets the X coordinate value.
- Y: This field sets the Y coordinate value.
- Z: This field sets the Z coordinate value.

3 Under Scale, you specify the scale for the inserted block. Specifying negative values for the X, Y, and Z scale factors inserts a mirror image of a block.

- Specify On-Screen: This option specifies the insertion point of the block using the pointing device.
- X: This field sets the X coordinate value.
- Y: This field sets the Y coordinate value.
- Z: This field sets the Z coordinate value.
- Uniform Scale: This option specifies a single scale value for X, Y, and Z coordinates. A value specified for X is also reflected in the Y and Z values.

4 Under Rotation, you specify the rotation angle for the inserted block.

- Specify On-Screen: This option specifies the rotation angle of the block using the pointing device.
- Angle: This field sets a rotation angle for the inserted block.

5 You can explode the block and inserts to the individual parts of the block. When you select Explode, you specify only an X scale factor.

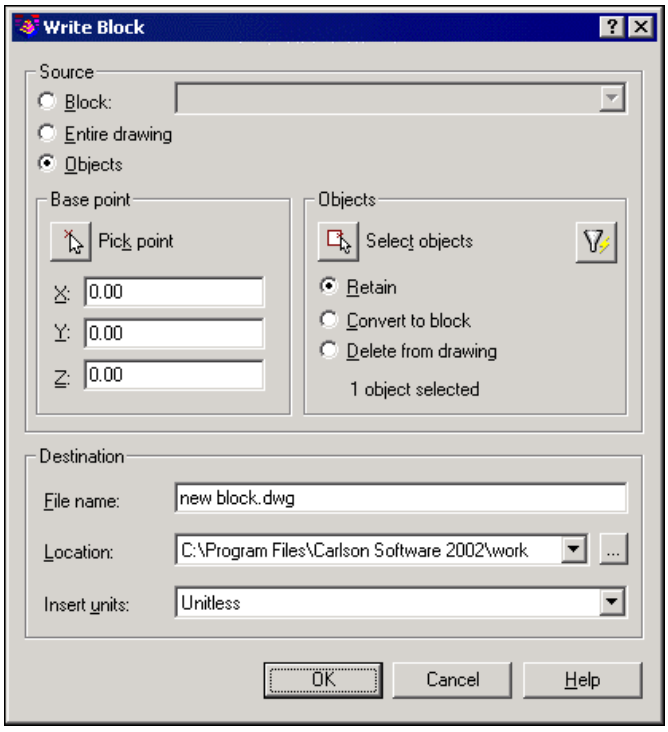
Prerequisite: None

Keyboard Command: DDINSERT

Write Block

Function

This command allows you to write objects or a block to a new drawing file.



The Write Block dialog box displays different default settings depending on whether nothing is selected, a single block is selected, or objects other than blocks are selected. For example, if you have a single block selected when you open the Write Block dialog box, the Source radio button is set to Block.

1 Under Source, you write selected blocks and objects out as a file, and specify insertion points.

- Block: This option specifies an existing block to save as a file. Select a name from the list.
- Entire Drawing: This option selects the current drawing as a block.
- Objects: This option specifies objects to be saved as a file.

2 Under Base Point, you must specify a base point for the block. The default value is 0,0,0.

- X: This field specifies the X coordinate value.
- Y: This field specifies the Y coordinate value.
- Z: This field specifies the Z coordinate value.
- Pick Point: This option allows you to temporarily close the dialog box so that you can specify an insertion base point in the current drawing.

3 Under Objects, you specify the objects to include in the new block and whether to retain or delete the selected objects or convert them to a block instance after you create the block.

- Retain: This option retains the selected objects as distinct objects in the drawing after you create the block.
- Convert to block: This option converts the selected objects to a block instance in the drawing after you create the block.
- Delete from drawing: This option deletes the selected objects from the drawing after you create the block.
- Select objects: This option dismisses the Block Definition dialog box temporarily while you select the objects for the block. When you finish selecting objects, press Enter to redisplay the Block Definition dialog box.
- Quick Select: This option displays the Quick Select dialog box, which defines a selection set.

- Objects Selected: This option displays the number of selected objects.

4 Under Destination, specify the name, location, and unit value used for the objects in the file.

- File Name: This field specifies a file name that the block or objects will be saved to.
- Location: This field specifies the drive and directory path for the file.
- Insert Units: This field specifies the unit value to be used when the new file is inserted as a block. Enter 0 (zero) if you do not want to scale the drawing to a specific value as you insert it.

Prerequisite: Drawing entities

Keyboard Command: WBLOCK

Text

Function

Creates a single-line text object.

You can use the TEXT to enter several lines of text that you can rotate, justify, and resize. As you type at the Enter Text prompt, the text you are typing is displayed on the screen. Each line of text is a separate object. To end a line and begin another, press Enter after entering characters at the Enter Text prompt. To end the TEXT command, press Enter without entering any characters at the Enter Text prompt.

By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle and immediately displays the Enter Text prompt. The text is placed directly beneath the previous line of text. The point specified at the prompt is also stored as the Insertion Point object snap.

Prompts

1 Current text style: "MONO" Text height: 4.00

2 Specify start point of text or [Justify/Style]: S

The style option lets you change the textstyle on the fly

3 Enter style name or [?] <MONO>: STANDARD

4 Current text style: "STANDARD" Text height: 4.00

5 Specify start point of text or [Justify/Style]: J

The justify option lets you specify the justification for the text.

6 Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: BC

In this case BC = Bottom Center

7 Specify bottom-center point of text: pick point or enter coordinates

8 Specify height <4.00>: press enter to select default or enter text height

9 Specify rotation angle of text <0d0'0">: press enter to select default or enter angle

10Enter text: Found Iron Pin

11Enter text: press enter to end

Prerequisite: None

Keyboard Command: DTEXT, TEXT

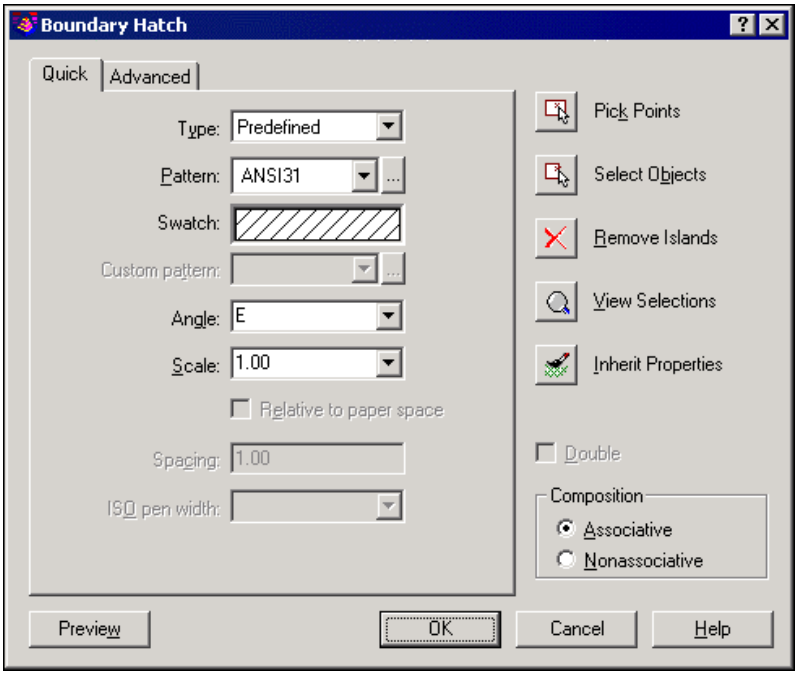
Hatch

Function

This command allows you to fill an enclosed area or selected objects with a hatch pattern.

The Hatch command first defines the boundaries of the area you want to hatch, either by computing a region or polyline boundary from a specified point within an enclosed area, or by using selected objects as boundaries. It then fills the boundaries with a hatch pattern or a solid color. You can create an associative hatch, which updates when its boundaries are modified, or a nonassociative hatch, which is independent of its boundaries. You can preview any hatch and adjust the definition.

Due to the large number of combinations of geometry that you can hatch, editing hatched geometry can produce unexpected results. In this event, delete the hatch object and rehatch.



The Boundary Hatch dialog box defines the boundary, pattern type, pattern properties, and attributes for hatch objects. Use the Quick Tab to work with hatch patterns and quickly create a hatch. Use the Advanced Tab to customize how TakeOff creates and hatches boundaries.

1 Under the Quick Tab you define the appearance of the hatch pattern to be applied.

- Type: This field sets the pattern type.
- Pattern: This field lists the available predefined patterns. The six most recently used predefined patterns appear at the top of the list. The Pattern option is available only if you set Type to Predefined.
- [...]: This button displays the Hatch Pattern Palette dialog box, in which you can view preview images for all predefined patterns at once to help you make a selection.

• Swatch: This field displays a preview of the selected pattern. You can click the swatch to display the Hatch Pattern Palette dialog box.

• Custom Pattern: This field lists the available custom patterns. The six most-recently used custom patterns appear at the top of the list. The Custom Pattern option is available only if you set Type to Custom.

• Angle: This field specifies an angle for the hatch pattern relative to the X axis of the current UCS.

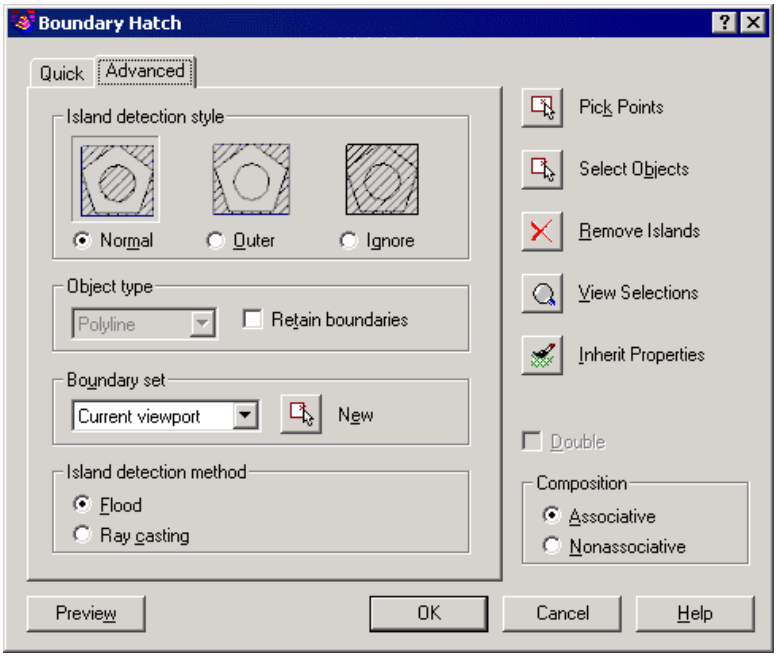
• Scale: This option expands or contracts a predefined or custom pattern. This option is available only if you set Type to Predefined or Custom.

• Relative to Paper Space: This option scales the hatch pattern relative to paper space units. Using this option, you can easily display hatch patterns at a scale that is appropriate for your layout. This option is available only from a layout.

• Spacing: This option specifies the spacing of lines in a user-defined pattern. This option is available only if you set Type to User Defined.

• ISO Pen Width: This option scales an ISO predefined pattern based on the pen width you choose. This option is available only if you set Type to Predefined and set Pattern to one of the available ISO patterns.

2 Under the Advanced Tab you define how TakeOff creates and hatches boundaries.



• Island Detection Style: This option allows you to specify the method for hatching objects within the outermost hatch boundary. If no internal boundaries exist, specifying an Island Detection style has no effect. Because you can define a precise set of boundaries, it's often best to use the Normal style.

The illustrations that accompany each style show how the program hatches a group of three nested boundary objects in each case.

Normal



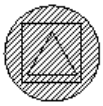
Hatches inward from the outer boundary. If the program encounters an internal intersection, it turns off hatching until it encounters another intersection. Thus, areas separated from the outside of the hatched area by an odd number of intersections are hatched, and areas separated by an even number of intersections are not.

Outer



Hatches inward from the outer boundary. The program turns hatching off if it encounters an internal intersection. Because this process starts from both ends of each hatch line, the program hatches only the outermost level of the structure and leaves the internal structure blank.

Ignore



Ignores all internal objects and hatches through them.

Hatching concave curves with the Outer and Ignore styles can cause hatching discrepancies.

The Normal, Outer, and Ignore options are also available from a shortcut menu by right-clicking in the drawing area while you specify points or select objects to define your boundaries.

• Object Type: This option allows you to specify whether to retain boundaries as objects, and specifies the object type TakeOff applies to those boundary objects. Object Type controls the type of the new boundary object. TakeOff creates the boundary as a region or a polyline. This option is available only if you select Retain Boundaries.

• Retain Boundaries: This option adds the temporary boundary objects to the drawing.

• Boundary Set: This field defines the set of objects TakeOff analyzes when defining a boundary from a specified point. The selected boundary set has no effect when you use Select Objects to define a boundary. By default, when you use Pick Points to define a boundary, the program analyzes all objects visible in the current viewport. By redefining the boundary set, you can disregard certain objects when defining boundaries without having to hide or remove those objects. For large drawings, redefining the boundary set can also produce the boundary faster because the program examines fewer objects.

• New: This option prompts you to select the objects that define the boundary set. When you choose this option, the dialog box temporarily closes, prompting you to select objects. TakeOff includes only the hatchable objects you select when it constructs the new boundary set. TakeOff discards any existing boundary set, replacing it with the new boundary set defined by the objects you select. If you don't select any hatchable objects, the program retains any current set. Until you exit the Hatch command or create a new boundary set, TakeOff ignores objects that do not exist in the boundary set when you define your boundaries using Pick Points.

• Island Detection Method: This option allows you to specify whether to include objects within the outermost boundary as boundary objects. These internal objects are known as islands.

• Flood: This option includes islands as boundary objects.

• Ray Casting: This option runs a line from the point you specify to the nearest object and then traces the boundary in a counterclockwise direction, thus excluding islands as boundary objects.

3 In the Boundary Hatch dialog box, you set the options the define the selection set.

• Pick Points: This option determines a boundary from existing objects that form an enclosed area. How TakeOff detects objects using this option depends on the selected Island Detection Method on the Advanced tab. For example,

if the Island Detection Method is Flood, the program detects objects within the outermost boundary as islands and includes them in the boundary definition. The Island Detection Style (which you also set on the Advanced tab) then determines how to hatch the detected islands. When you choose Pick Points, the dialog box closes temporarily, and the program prompts for point specification.

• Select Objects: This option allows you to select specific objects for hatching. The dialog box closes temporarily, and the program prompts you for object selection. When you define your boundaries using Select Objects, the program does not detect interior objects automatically. You must select the objects within the selected boundary to hatch those objects according to the current Island Detection Style (which you set on the Advanced tab). Each time you choose Select Objects, the program clears the previous selection set. While selecting objects, you can right-click at any time in the drawing area to display a shortcut menu. You can undo the last or all selections, change the selection method, change the island detection style, or preview the hatch.

• Remove Islands: This option removes from the boundary definition any of the objects that the program detects as islands when you use Pick Points. You cannot remove the outer boundary.

• View Selections: This option temporarily dismisses the dialog box and displays the currently defined boundaries with the hatch settings that you last previewed. This option is unavailable when you have not yet specified points or selected objects.

• Inherit Properties: This option hatches specified boundaries using the hatch properties of one object. After selecting the associative hatch object whose properties you want the hatch to inherit, you can right-click in the drawing area and use the shortcut menu to toggle between the Select Objects and Pick Internal Point options to create boundaries.

• Double: For user-defined patterns, this option draws a second set of lines positioned at 90 degrees to the original lines, creating a crosshatch. This option is available only if you set Type to User Defined on the Quick tab.

• Associative: This option creates an associative hatch, meaning that the hatch is updated when you modify its boundaries.

• Nonassociative: This option creates a nonassociative hatch, meaning that it is independent of its boundaries.

• Preview: This option temporarily dismisses the dialog box and displays the currently defined boundaries with the current hatch settings. This option is not available when you have not yet specified points or selected objects to define your boundaries.

Prerequisite: None

Keyboard Command: BHATCH

2 Tangents Radius

Function

This command fits a curve between two tangent lines using a known radius. The command prompts for the radius and the points on the two tangent lines.

Prompts

1 Radius of Arc <0.0>: enter radius value

2 [nea] Pick Point on 1st Tangent Line: pick a point

3 [nea] Pick Point on 2nd Tangent Line: pick a point

Menu Location: Draw, Curves >

Prerequisite: Tangent lines should be drawn before execution

Keyboard Command: 2TANLIN

2 Tangents Arc Length

Function

This command fits a curve between two tangent lines and a known arc length. It prompts for the arc length, the P.I. (point of intersection of the tangent lines), and the points on the two tangent lines.

Prompts

1 Arc Length <0.0>: enter or pick distance

2 [int on] Pick P.I. of curve: pick intersection of tangent lines

3 [nea on] Pick point on 1st Tangent Line: pick point

4 [nea on] Pick point on 2nd Tangent Line: pick point

Prerequisite: Tangent lines should be drawn before execution.

Keyboard Command: 2TANLAL

2 Tangents Chord Length

Function

This command fits a curve between two tangent lines and a known chord length. It prompts for the chord length, the P.I. (point of intersection of the tangent lines), and points on the two tangent lines.

Prompts

1 Chord Length <0.0>: enter or pick distance

2 [int on] Pick P.I. of curve: pick intersection of tangent lines

3 [nea on] Pick point on 1st Tangent Line: pick point

4 [nea on] Pick point on 2nd Tangent Line: pick point

Prerequisite: Tangent lines should be drawn before execution.

Keyboard Command: 2TANLCL

3 Point Curve

Function

This command draws an arc between three points. The first point is the PC, the second is a point on the arc, and the third is the PT. The points can either be picked on-screen or specified by point number.

Prompts

1 Pick PC point or point numbers: 101

2 Pick Second point or point number: 102

3 Pick PT point or point number: 103

Prerequisite: None

Keyboard Command: 3PA

PC PT Radius Point

Function

This command draws an arc between the PC point, radius point, and PT point. The points can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you want the arc to go in the other direction, enter No.

Prompts

1 Pick PC point or point number: 101

2 Pick Radius point or point number: 102

3 Pick PT point or point number: 103

4 Is the direction of this arc correct ? No/<Yes>: N

The arc draws in the other direction.

Prerequisite: None

Keyboard Command: PCA

PC Radius Chord

Function

This command draws an arc given the PC point, radius length, chord length, and chord bearing. The PC point can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you want the arc to go in the other direction, enter No.

Prompts

1 Radius of Arc <-40.00>: 500

2 PC Start Point ?

Pick point or point number: pick a point

3 Chord bearing or chord endpoint (<Bearing>/Point)? Press Enter

4 Enter Bearing (Qdd.mmss) <90.0000>: 145.1041 (for NE 45°10'41")

5 Chord Length <200.46>: 200

6 Is this arc in the correct direction (<Yes>/No)? Press Enter

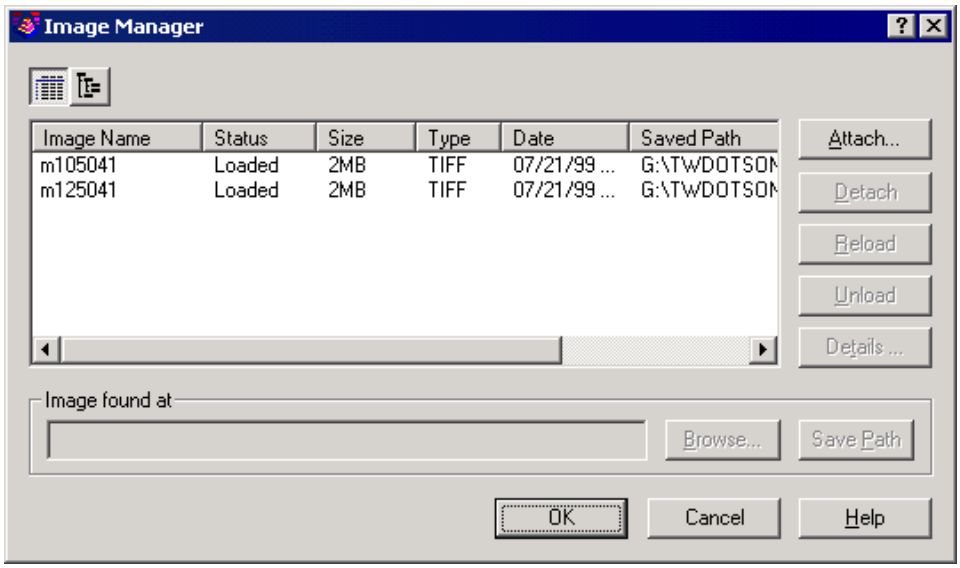
Prerequisite: None

Keyboard Command: SRCB

Raster Image

Function

This command allows you to manage raster images.



1 The Image Manager dialog box lists all the image files attached to the current drawing. You can view the parameters and details for selected images. You can attach new image files and detach, locate, reload, and unload existing images.

• List View: This button lists the image definitions attached to the drawing. Each image name appears only once regardless of how many times you attach (insert) the image. You can sort the list of images by name, status (loaded, unloaded, or not found), size, type (TIFF, for example), date, or the saved path and file name. By default, TakeOff displays the list alphabetically by image name.

To select multiple images, hold down SHIFT or CTRL while selecting items.

To sort the list alphabetically or numerically by a specific column, click that column's heading.

To change the width of the column, drag the line between the column headings to the right or left. The program saves and restores the settings when you reopen the dialog box.

To change an image name, select it and then click it again, or select it and then press F2. You cannot edit names of images that reside in external references (xrefs). Image names can include up to 255 characters and can contain letters, digits, spaces, and any special characters not used by Microsoft® Windows® or TakeOff. The image name can be identical to the file name, but changing the image name does not change the file name.

• Tree View: This button displays all the image definitions and the levels of nesting of images within xrefs. The top level of the tree view shows images that you attached directly to the drawing, images nested in block references, and the names of externally referenced drawings containing images. The names of the images attached to the externally referenced drawings appear nested within the drawing at the next tree level. To insert a copy of an already attached image, select it, and then choose Attach.

Tree view lists the image names only (not file names) and lists the image name just once, regardless of how many times you attach (insert) the image.

You can edit an image name by selecting it and then clicking it again, or by selecting it and then pressing F2. However, you cannot select more than one image at a time.

• Attach: This option displays the Select Image File dialog box. When you unload and then reload an image, the program draws that image on top. Images remain loaded or unloaded from one drawing session to the next.

• Detach: This option removes the selected image definitions from the drawing database and erases all the associated image objects from the drawing and from the display.

• Reload: This option loads the most recent version of an image or reloads an image that was previously unloaded. Reloading does not control whether the image is displayed, but it ensures display of the most current image.

• Unload: This option unloads image data from working memory without erasing the image objects from the drawing. It is recommended that you unload images no longer needed for editing to improve performance. An unloaded image cannot be displayed or plotted. You can selectively load and unload individual images from a working list of images associated with the drawing file.

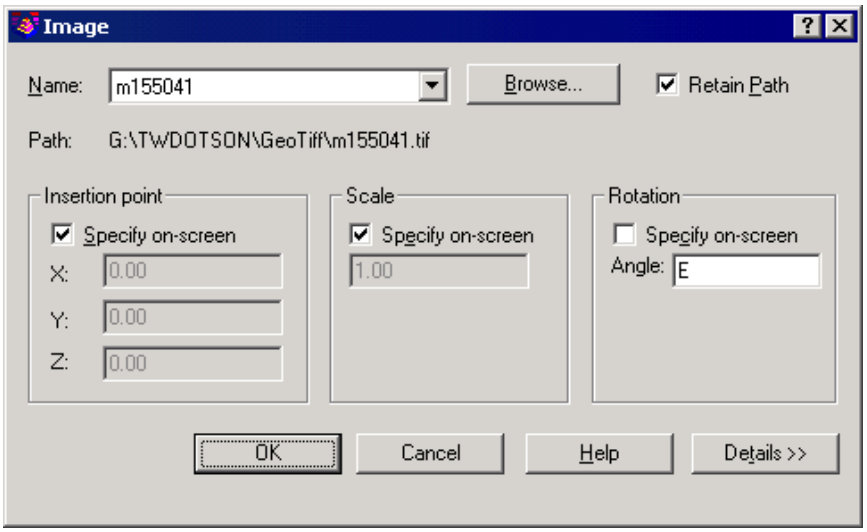
• Details: This option opens the Image File Details dialog box, which displays the image name, saved path, active path, file creation date and time, file size and type, color system, color depth, width and height in pixels, resolution, default size in units, and a preview image.

• Image Found At: This field shows the path of the selected image. If you select multiple images, this field remains blank. The path shown is the actual path where the image resides.

• Browse: This option opens the Select Image File dialog box (a standard file selection dialog box). The path you select appears under Image Found At.

• Save Path: This option stores the new path information. Press ESC while editing the path to restore the old path. If the program cannot find the referenced image in the new path, the image's status changes to Not Found. If you do not choose Save Path after editing the path, the program uses the original image path the next time you load the drawing.

2 Under the Image dialog box, you can attach an image.



3 In the Image dialog box, you must first identify the image and the path.

• Name: This field identifies the image you have selected to attach, either from the Select Image File dialog box (an unattached image) or from the list of previously attached images. To add another instance of an image file that

is already attached, select the image name from the list and choose OK.

• Browse: This option opens the Select Image File dialog box (a standard file selection dialog box). If Show Preview is selected, the program displays a preview of the selected file.

• Retain Path: This option saves the path of the image file with the image definition. If Retain Path is not selected, only the image name is saved and TakeOff searches the Support File Search Path.

4 Under Insertion Point, you must specify the insertion point for the selected image. Specify On-Screen is the default. The default insertion point is 0,0.

• Specify On-Screen: This option directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.

• X: This field sets the X coordinate value.

• Y: This field sets the Y coordinate value.

• Z: This field sets the Z coordinate value.

5 Under Scale, you must specify the scale factor of the selected image. Specify On-Screen directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.

6 Under Rotation, you must specify the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value on the command line. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

Prerequisite: Raster image

Keyboard Command: IMAGE

Place Image by World File

Function

This function allows you to insert Geo-Referenced TIF files into AutoCAD drawings. This process requires the presence of an accompanying TFW file. The TFW file contains information about the location and scaling of the actual raster image TIF file. This eliminates the guesswork in inserting, moving, and rotating raster images to the project area. You begin by selecting the TFW or JGW file to process. If the related TIF file is present in the same directory, the image will be inserted into the proper coordinates.

Prompts

Select World File: *choose existing .TFW or .JGW file*

Keyboard Command: geotiff
Prerequisite: None

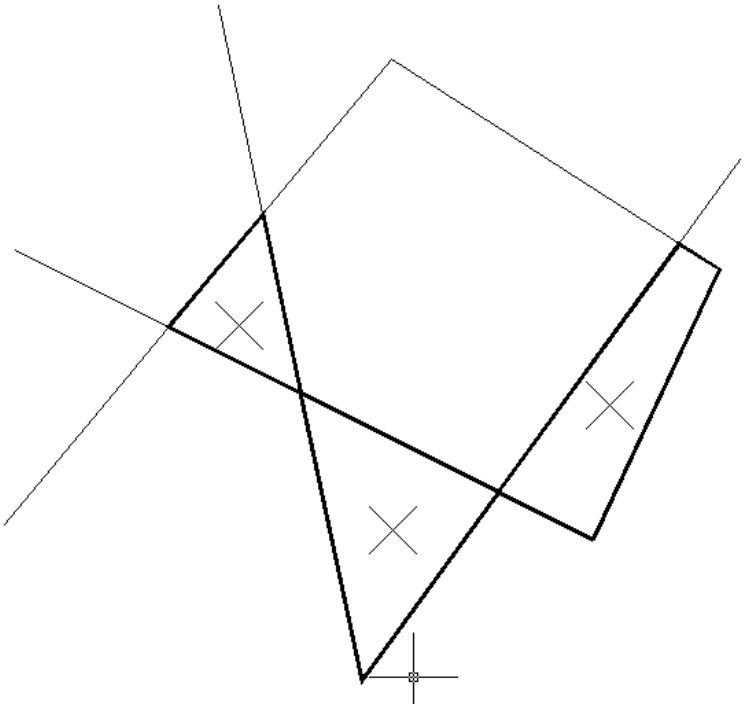
Closed Polyline By Interior Point

Function

To create closed polylines from any existing linework, simply select all entities you would like to use and specify desired snap tolerance. Then click inside openings you would like to trace and the routine will generate corresponding closed polylines. The duplicate polylines are detected and not created, so that clicking more than once in the same area does not change anything. These new polylines are always created in the current layer. Layers of the original linework do not matter.

Prompts

Select polylines: *pick an entities to be used*
Enter snap tolerance or press Enter for none:
Pick an internal point: *pick the points to enclose*



These three polylines are created from original linework by clicking at shown locations

Keyboard Command: boundpl
Prerequisite: Entities

ShrinkWrap Entities

Function

This command allows you to create a closed polyline that encloses a given set of entities. The resulting polyline is created on the current layer. The command works on either point entities or polylines. For points, the command creates a closed polyline through the points around the perimeter of the area defined by the points. For polylines, the shrink-wrap polyline follows the outside border of the selected polylines. The polylines that are processed must be connected to be shrink-wrapped. The snap tolerance is the maximum gap that will be joined to make the closed polyline.

Prompts

1 Entities to shrink wrap [Points/<Linework>]? P

Select the points to shrink-wrap.

2 Select objects: select points

3 Select objects: press enter to continue

Reading points ... 205

Triangulating points ... 205

4 Create 2D or 3D Polyline (<2D>/3D)? 3D

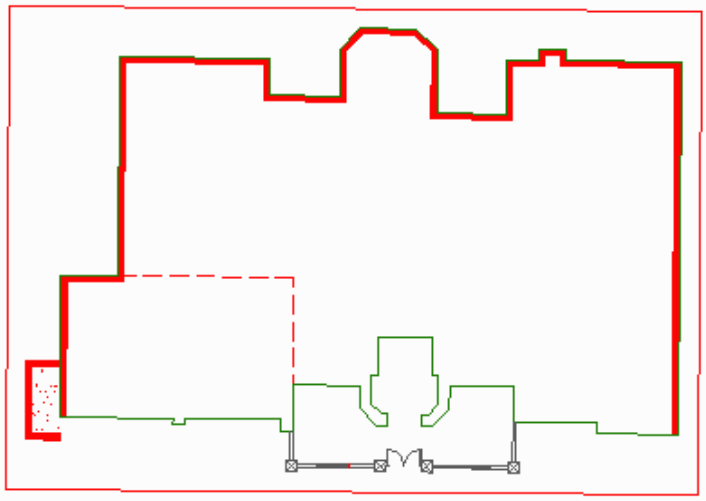
Prerequisite: Entities on screen.

Keyboard Command: SWPLINES

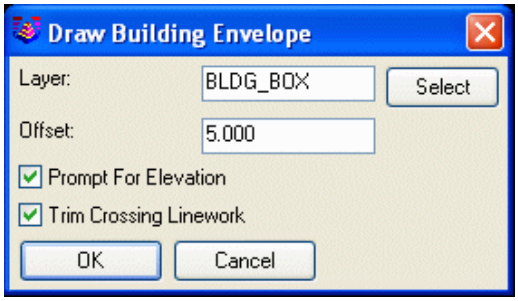
Building Envelope Polyline

Function

This command creates a rectangler polyline around selected linework. This can be used to give a building all one elevation.



Select the entities that make up the building. Next you will be prompted to name the layer and to set an offset distance (The above example is offset by 5 feet). Also, you can set the elevation of the envelope and trim crossing linework to ensure you have a flat pad.



Prompts

Draw Building Envelope dialog
Select building lines.
Select objects: *pick the linework that makes up the perimeter of the building*
Draw another building envelope [<Yes>/No]? *N*

Prerequisite: a pad

Keyboard Command: bldg_perim

Design Pad Template

Function

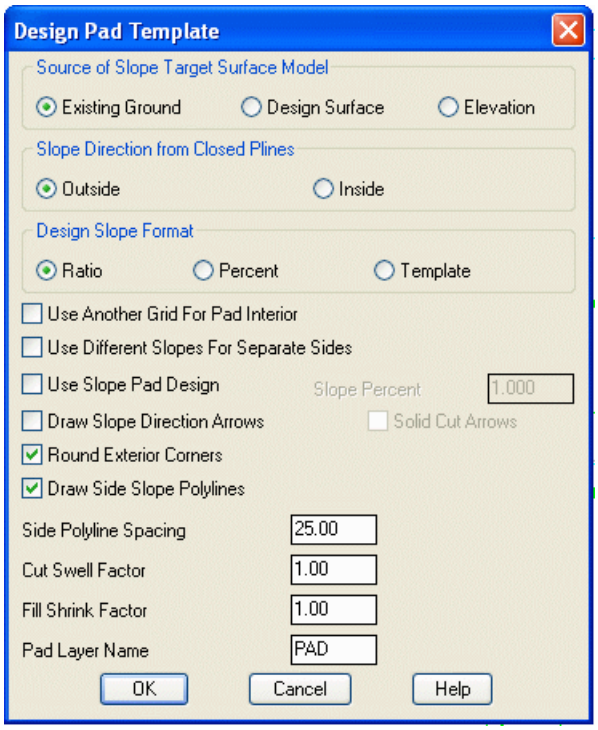
This command creates design slopes from a perimeter polyline at specified cut/fill slopes to reach existing ground. This routine can be used to design building pads, pits, roads, ditches, stockpiles, etc. The design is drawn as 3D polylines for the cut/fill slopes and for the daylight perimeter where the design meets existing ground. Before beginning this routine, you must have drawn the polyline perimeter. The polyline perimeter can be either a 2D or 3D closed or open polyline. For a 2D polyline, the program will prompt for an elevation for the pad perimeter. With a 3D polyline, the pad perimeter is set to the elevations of the 3D polyline. For an open polyline, the program will prompt for the side for the design. With a closed polyline, the program designs the slopes either outward or inward depending on the setting in the dialog.

Source of Surface Model: Choose between a surface file, screen entities, or a flat elevation.

Slope Direction from Closed Polylines: For closed pad perimeters, there is an option to draw the slopes inward or outward from the perimeter. The outward method starts the slopes at the design elevation of the perimeter and projects out to intersect the existing surface. The inward method projects the slopes inside to reach the grid surface or a set elevation.

Design Slope Formula: Choose between Slope, Ratio, or Template. The template (.TPL) file is created in the *Design Template* routine in the Section Profile module. When using a template, the pad perimeter represents the centerline. One way to create the pad perimeter for the template
is to use the *Profile to 3D Polyline* command which converts a 2D centerline to a 3D polyline using a design profile. With a template, the program uses not only the cut and fill slopes from the template file but also draws all the template grade points such as edge of road, curb and ditch. The subgrade, superelevation and template transition options of the template file are not used in this command. These options are only applied in the *Process Road Design*

command. The grade points are drawn as 3D polylines parallel with the centerline. Cross section 3D polylines that include the grade points are also drawn at the specified interval.



Use Another Grid for Pad Interior: This option will prompt for another grid (.GRD) file to use for the design surface within the starting pad perimeter. Otherwise the program will model the pad interior by straight interpolation from the starting pad perimeter elevations. For example, if a building pad has a starting pad perimeter at a set elevation and the pad is supposed to be flat, then this option is not needed. This option is needed in a case where you are designing a pit and the starting pad perimeter is a 3D polyline that follows an undulating pit bottom surface. The pad design will model the pit side slopes. In order to model the undulating bottom of the pit, you need the Use Another Grid for Pad Interior option to select a grid that models the pit bottom.

Use Different Slopes for Separate Sides: Allows you to specify different slopes for different sides of your pad polyline. If this is toggled ON, you will be prompted to specify two of more slope groups, then you will be prompted to select the polyline segments that belong to each group.

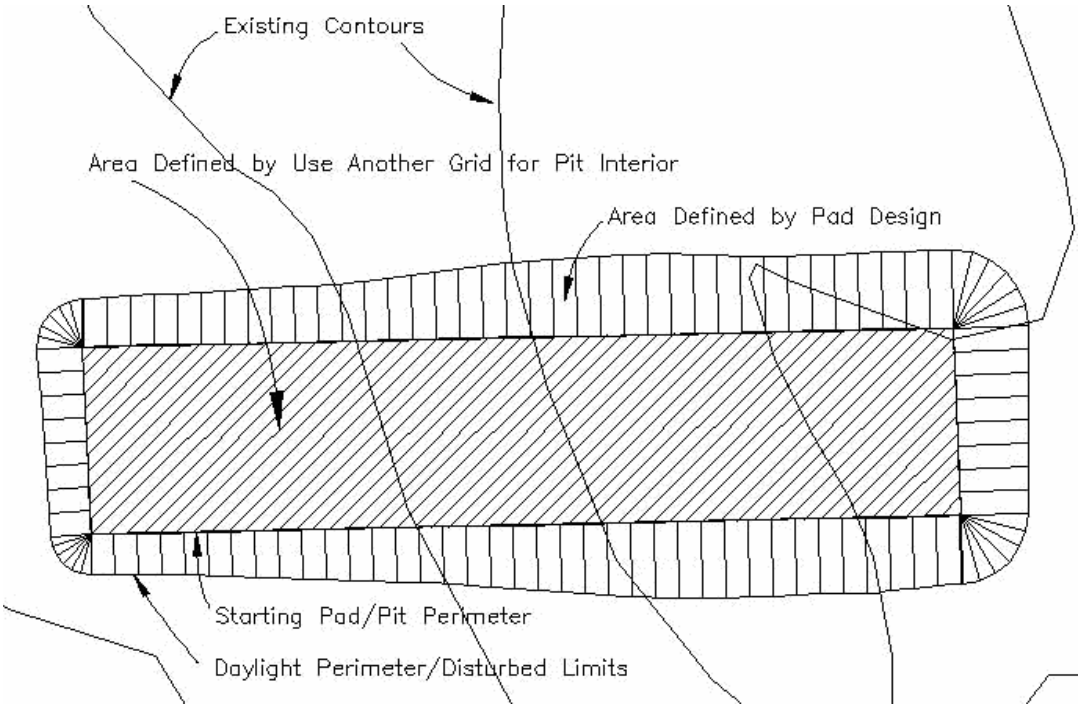
Use Slope Pad Design: Allows you to set a cross slope amount for the top of the pad. You will be prompted to screen pick two points that designate the slope direction. For automatic balancing of cut/fill quantities, you will be prompted to find the optimal slope and slope direction.

Draw Slope Direction Arrows: This option draws an arrow on the outslopes that points in the downhill direction. Arrows on fill slopes are drawn as solid filled.

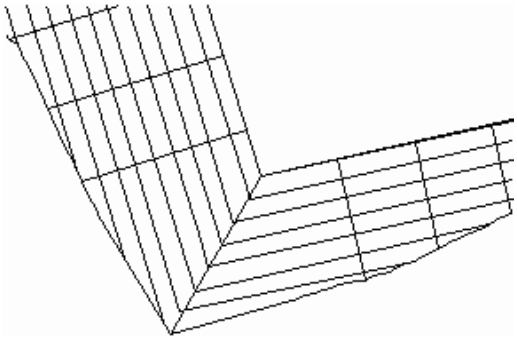
Solid Cut Arrows: Allows you to choose between drawing the cut arrows as solid filled or as wire frame.

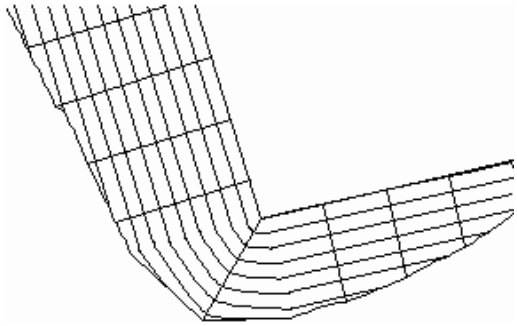
Round Exterior Corners: This option holds the outslopes around the corners. Otherwise the side outslopes stay straight until they meet at the corners as shown in the figure.

Draw Side Slope Polylines: When this option is ON, Design Pad Template will draw 3D polylines perpendicular to the pad perimeter from the pad to the catch point.


Existing Contours
Area Defined by Use Another Grid for Pit Interior
Area Defined by Pad Design
Starting Pad/Pit Perimeter
Daylight Perimeter/Disturbed Limits

Example of pit design for option of Use Another Grid for Pad Interior





Pad corner with round corners option.
Pad corner without round corners option.
Side Polyline Spacing: Specifies the interval at which to draw the Side Slope Polylines. Besides at the interval, side slope polylines are also drawn at grid corners.

Cut Swell Factor: Cut volume is multiplied by this factor in the final volume report.

Fill Shrink Factor: Fill volume is multiplied by this factor in the final volume report.

Pad Layer Name: You must specify the layer name that the pad 3D polylines will be drawn on.

There is an option to calculate volumes for the pad design. The volumes are calculated by comparing the existing surface with the pad design. The inclusion perimeter for the volume calculation is the daylight perimeter polyline which represents the limits of disturbed area. The existing surface model is defined by the existing grid (.GRD) file or screen entities selected at the beginning of the command. The pad design surface is calculated by making a grid from the pad 3D polylines including the starting pad perimeter, the side polylines and the daylight perimeter.

Besides calculating the volumes in the *Design Pad Template* routine, you can also calculate the volumes with the *Two Surface Volumes* command. The design surface for *Two Surface Volumes* can be the final output grid from Design Pad or you can create a design grid with *Make 3D Grid File* using the 3D polylines created in Design Pad. The reasons to use *Two Surface Volumes* are that this volume routine has more output options (cut/fill color maps, etc.) and you can check the volumes by plotting or contouring the grid files. Also you can combine several pads and other final surfaces by running *Make 3D Grid File* and then calculate the overall site volumes with *Two Surface Volumes*.

The surface is defined by a 3D rectangular grid mesh. The grid can be selected from a grid (.GRD) file or calculated by the routine. When calculated, *Design Pad Template* internally creates a grid mesh from the surface entities found by doing a crossing selection of the grid limit. Make sure that the grid area covers the entire area for the pad including room for the cut/fill slopes.

The design is drawn as 3D polylines and the earthwork volumes are calculated. Before ending, the program allows you to adjust the design by changing the pad elevation, slopes and offset. The program can find the cut/fill balance by automatically adjusting the pad elevation. If adjustments are specified, the pad polylines are redrawn and the volumes recalculated. At the end, there is an

option to trim existing contours inside the disturbed limits of the pad. Then there is an option to draw contours on the pad. If contouring is selected, a dialog lets you set the contouring options. Usually you should specify a new contour layer and turn off smoothing.

The cut and fill slopes can be defined by ratio, percent slope or template file. Using a template file lets you do road design without cross sections. Instead the existing surface is modeled by the grid. To do road design by cross section, use the *Process Road Design* command in the Section Profile module.

Prompts

Pick Lower Left limit of pad disturbed area: *Pick lower left* These prompts appear for the Screen Entities surface model method.

Pick Upper Right limit of pad disturbed area: *Pick upper right*

Be sure to pick these limits well beyond the area of the top of pad polyline in order to make room for the outslopes.

Make Grid File Dialog

After selecting the limits of the disturbed area the program will generate a 3D grid that represents the surface. Specify the grid resolution desired and select OK.

Pick the top of pad polyline: *Select perimeter polyline*

Enter the fill outslope ratio <2.0>: *2.5*

Enter the cut outslope ratio <2.0>: *2.5*

After entering outslopes slope ratios, a range of elevations along the pad top will be noted.

Enter the pad perimeter elevation <29.54>: *39*

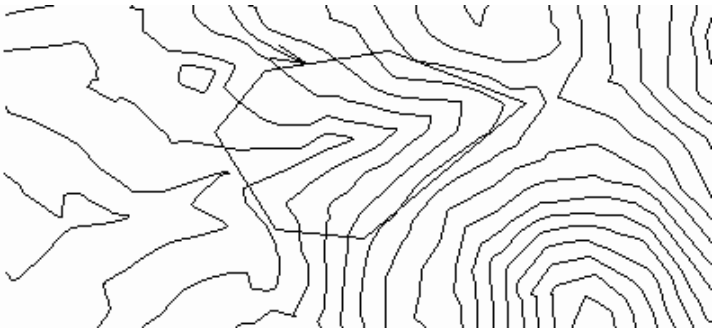
Calculate earthwork volumes (<Yes>/No)? *Press Enter*

Report Viewer Reports cut/fill volume

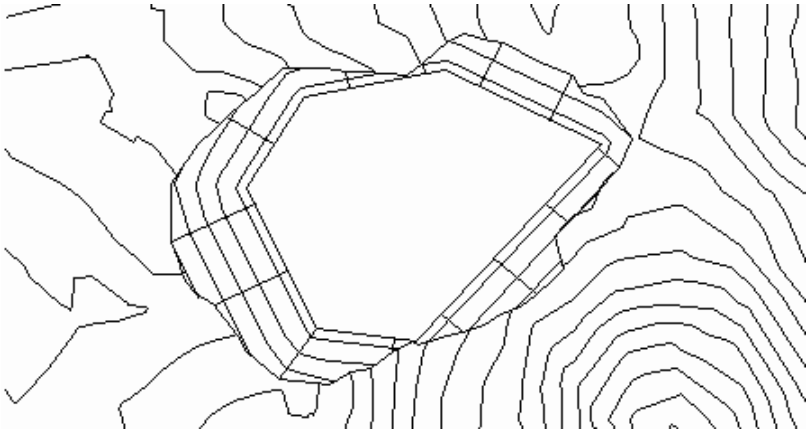
Write final surface to grid file (Yes/<No>)? *Press Enter.* This option will output a grid file using the elevations of the pad within the disturbed area polyline and using the original ground surface everywhere else.

Adjust parameters and redesign pond (Yes/<No>)? *Press Enter*

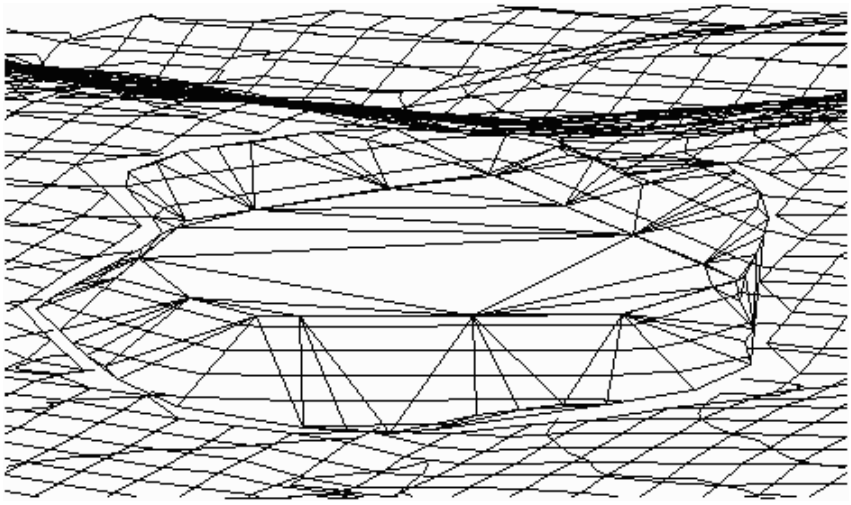
Update surface file [Yes/<No>]? Press Enter. This option will merge the new pad .TIN file with the rest of the design surface.



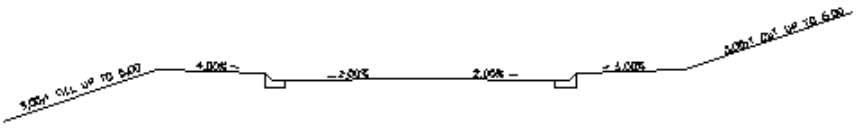
Existing contours with top of pad perimeter polyline



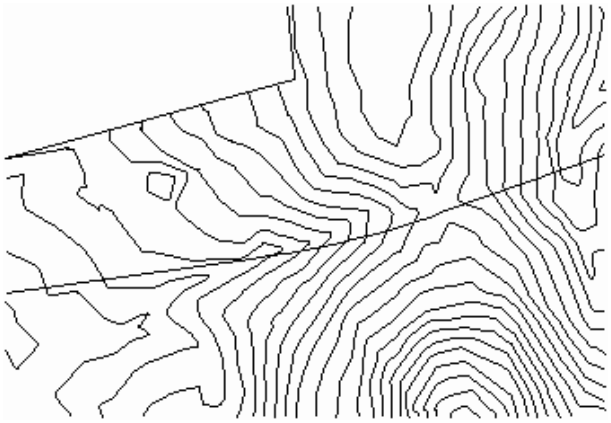
Pad template with contours



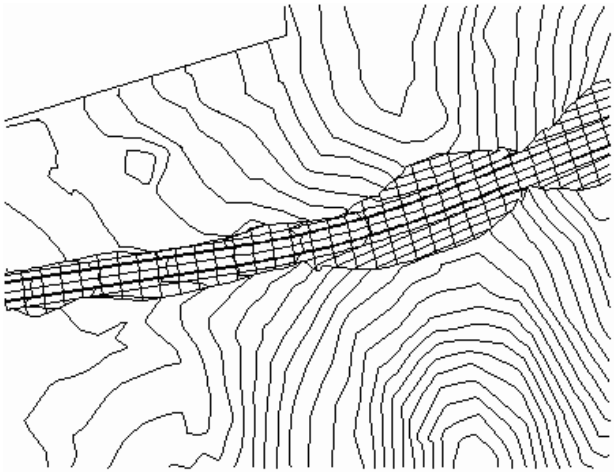
3D view of pad with DTM of surface and triangulation faces of pad



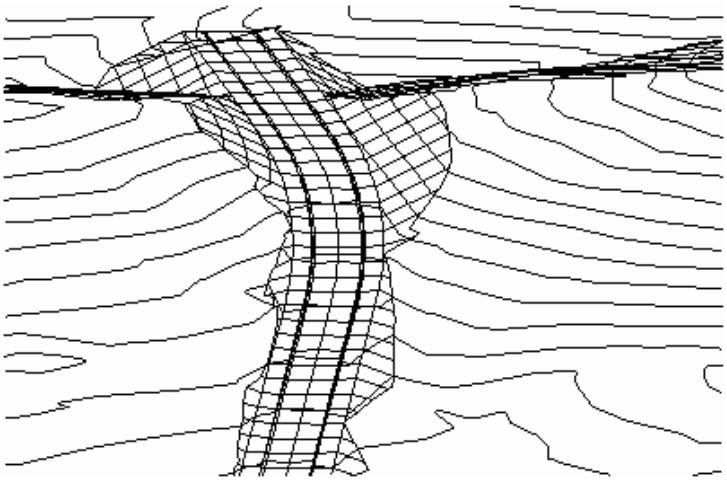
Template to apply in Design Pad Template



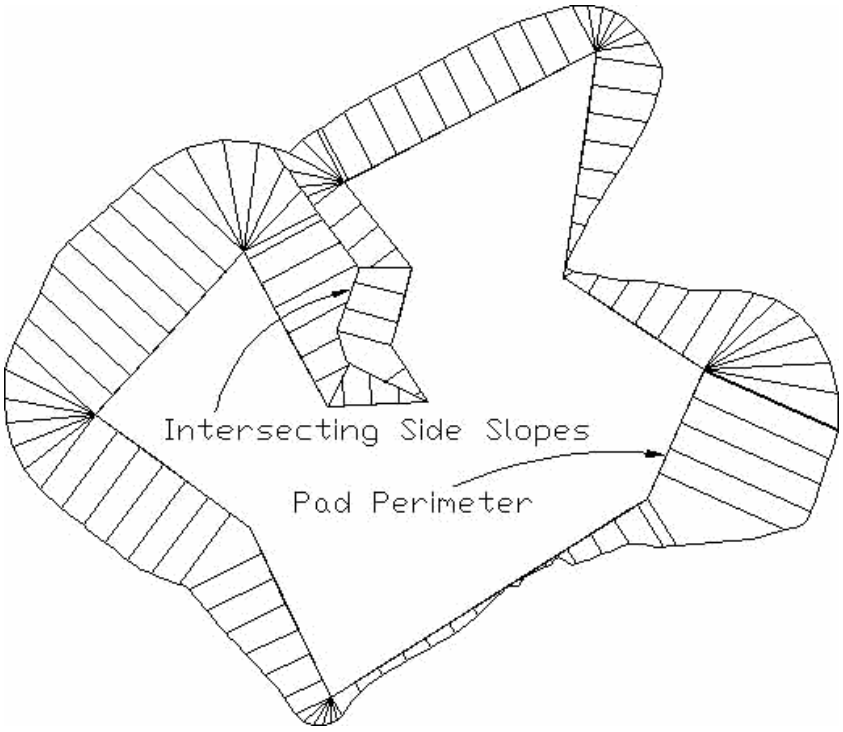
Existing surface with 3D polyline centerline



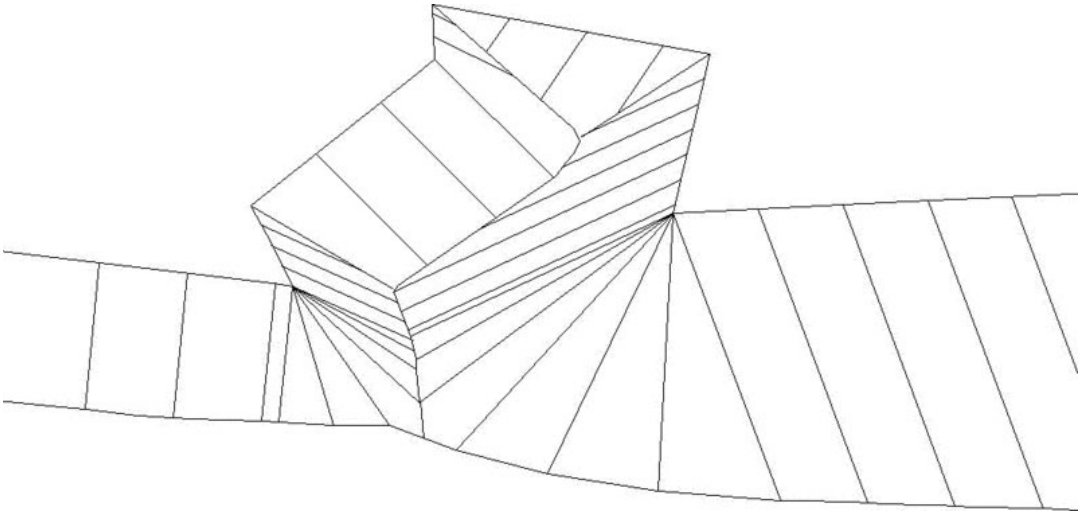
Result of Design Pad Template showing template grade polylines, cross section polylines, cut/fill slopes, and final contours.



Viewpoint 3D view of Design Pad Template



Design Pad Template can also handle self-intersecting side slopes



Viewpoint 3D view of intersecting side slopes

Prerequisite: A pad perimeter polyline and surface entities or a surface file

Keyboard Command: pad

Title Block

This function is different depending on your AutoCAD version

Function (AutoCAD R14)

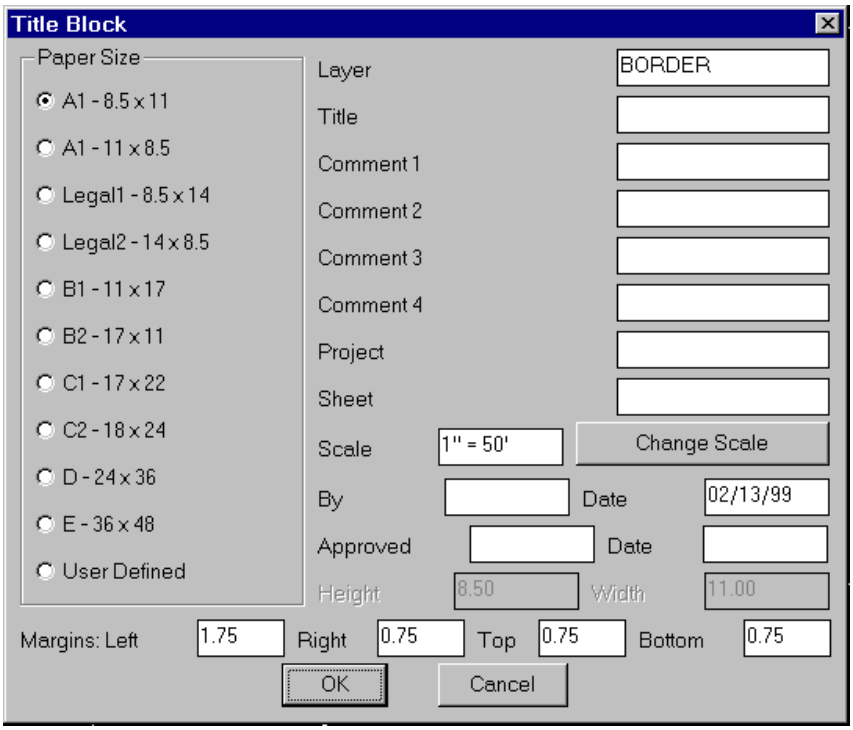
This command draws a border and title block for the selected sheet size. The margins to useare specified at the bottom of the dialog. Margins are needed so that the border fits in theplotter's plotable area. For sheet 11x17 or smaller, a 1/2 inch margin is typical. For larger sheets, atypical margin is 3/4 inch. The *LIMITS* of the drawing can be set to the lower left and upper rightcorners of the border. To change the title block, edit the drawing TBLOCK.DWG in the \SUP directory.Click the toggle User Defined to set a custom sheet size. The default user defined size can be storedin the Configure TakOff command under the General Settings option. After the title blockis drawn, the contents can be edited using the Attribute Edit command under the Modify menu.The Change Scale button will change the scale for the title block and for the drawing.

Function (AutoCAD R2000 and up)

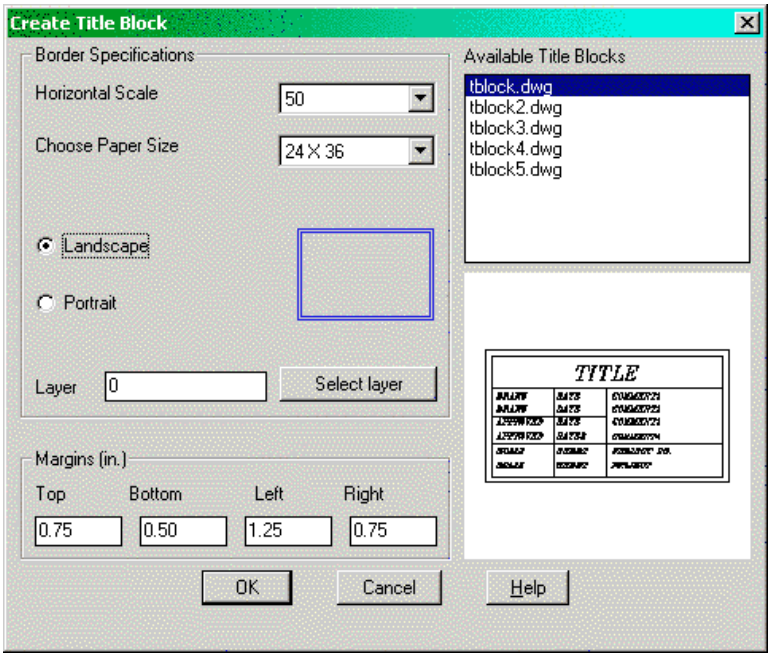
This command draws a border and title block for the selected sheet size. At the top of thedialog, choose your horizontal scale and sheet size. The *other* choice at the bottom of each list willallow you to add your own scale or size if yours is not listed. Anything added to these lists will beretained for future use. Next, choose either "landscape" or "portrait" format. A blue rectangle next tothis choice shows you the difference. Below this, you can choose what layer to draw the borderand title block on. The margins to use are specified next at the bottom of the dialog. On the righthand side of the dialog. you can choose from several title blocks. As you choose each one, apreview will be shown below this list. This routine looks for all drawings named "tblock" in the\SUP directory. If you want to add your own title block, simply create a new drawing (or copy anexisting one) in the \SUP directory and give it a name that starts with tblock. Example: tblock22.dwgand tblock-Jones.dwg are both valid names for this routine, but "MyTitleblk.dwg" is not. Afteryou have made all your decisions in the dialog box, press OK. Depending on your current zoomlevel, your drawing may be zoomed out to allow you to see the entire area that will be covered bythe drawing border. At this point, you have the border attached to your cursor and itis waiting for you to pick a point for insertion. As soon as you do this, a secondarydialog will appear for you to fill out theattributes associated with the particular title blockyou selected.

Prerequisite: set horizontal scale in Drawing Setup

<>**Keyboard Command:** tblock



Title Block Dialog for R14



Title Block Dialog for R2000 & up</>

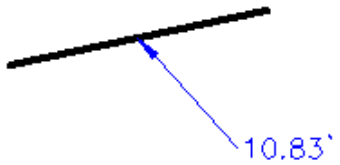
Distance with Leader

Function

This command labels the distance of a line or polyline segment at a point then draws a user specified leader line to point to the defining line. There is the ability for multi-segment leaders, and the option to align the label horizontal to the current view or parallel to the linework.

Prompts

Define distance by, Points/<Select line or polyline>: *select a line*
Pick point to start leader: *pick a point near the line*
Label Position: *pick a point*
Define distance by, Points/<select line or polyline>: *press Enter to end*



Keyboard Command: distlead
Prerequisite: None

Curve_Arrow

Function

Curve - Arrow can be used to draw a section of contour line or create leader pointer lines. Curve - Arrow draws a Bezier curve through user specified points. After choosing endpoints, each time an intermediate points is picked the curve will be redrawn through all the points. There is an option to draw an arrowhead at the starting point. The arrowhead size is determined by the AutoCAD system variable "DIMASZ". In order to change this size, type DIMASZ at the AutoCAD command prompt. This routine also has a Zorro option which creates a Z leader curve.

Prompts

Do the zorro (Yes/<No>)? *N*
Include an arrow (Yes/<No>)? *Y*
Enter the arrow head size <4.00>: *press Enter* This defaults to the DIMASZ system variable.
Pick a starting point: *pick a point*
Pick an ending point: *pick a point*
Pick an intermediate point (U to Undo): *pick a point*
Pick an intermediate point (U to Undo): *press Enter*

Keyboard Command: carrow

Prerequisite: None

Bar Scale

Function

This command draws a barscale. You will be prompted for the horizontal scale. The default value is set in the Drawing Setup command in the Settings menu.



Prompts

1 Insertion Point: **pick a point**

2 Horizontal scale <50.0>: **Press Enter**

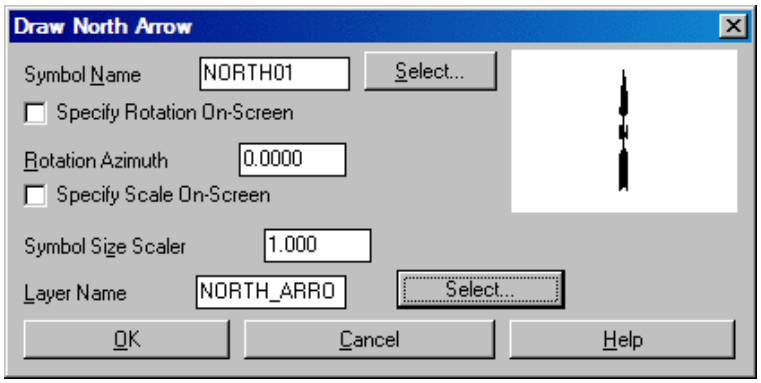
Prerequisite: None

Keyboard Command: BARSCALE

North Arrow

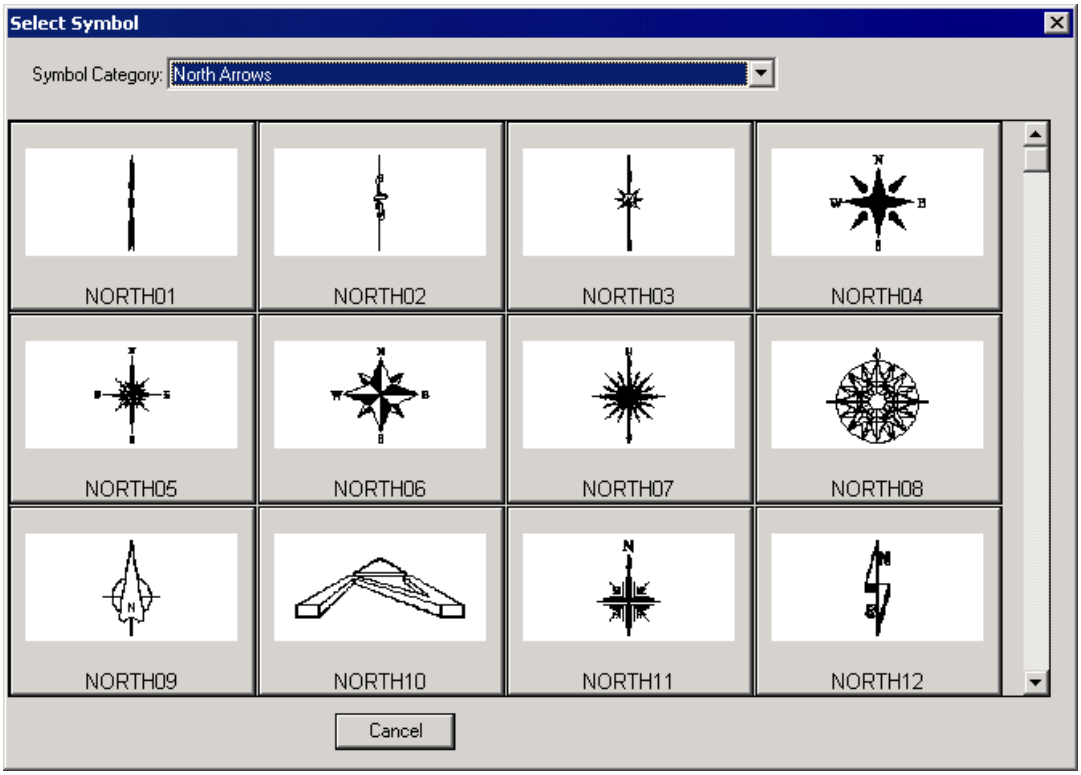
Function

This command inserts a north arrow symbol. You can select from several styles of arrows, and you can add your own by using the *Edit Symbol Library* command on the Settings menu.



Prompts

Draw North Arrow Dialog *choose an arrow symbol, layer and other variables*
Specify insertion point: *pick a point*



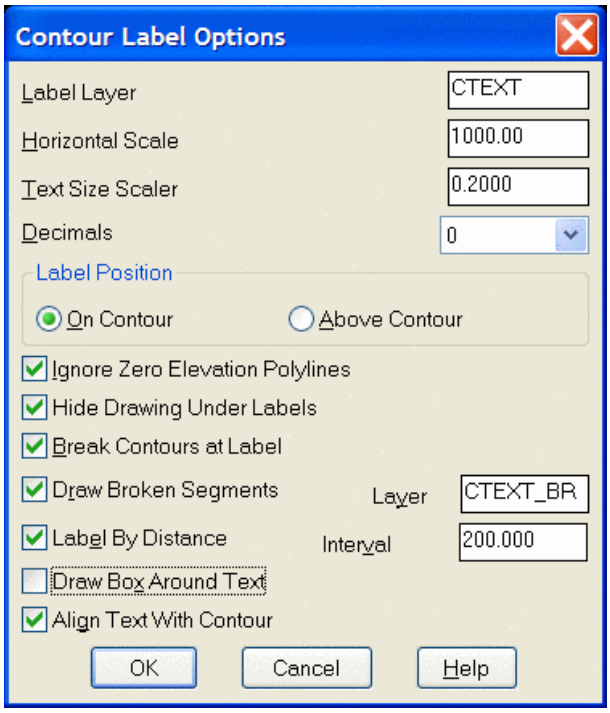
Keyboard Command: narrow
Prerequisite: None

Contour Elevation Label

Function

This command can be used to simultaneously create elevation labels on a group or groups of contour polylines at elevation. To place the labels, pick two points crossing the contour polylines at the desired label location. The program will find all the contour polylines that intersect the picked line (defined by the two picked points) and will place labels at the intersection point of each contour. A second crossing line can be initiated immediately, so multiple areas can be quickly labeled while remaining in the command. The actual "z" elevation of the contour line determines the label value.

Contour Label Options Dialog



Label Layer specifies layer name for the contour labels that will be created.

Horizontal Scale is used in conjunction with the Text Size Scaler to determine unit height of the contour labels.

Text Size Scaler is a scaler that will be multiplied by the horizontal scale to set the actual text height of the labels in AutoCAD units.

Decimals sets the decimal precision for the labels to be created.

Label Position determines the label position in relation to the contour polyline.

- **On Contour** centers the label on the contour line.
- **Above Contour** places the label above the contour line. If this option is used, the options for Break Contours at Label and Draw Broken Segments become inactive.

Ignore Zero Elevation Polylines enables the routine to filter out all entities with an elevation of zero.

Hide Drawing Under Labels activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour line is still fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, yet the contour lines themselves remain contiguous. This feature will also hide other entities that are in the immediate vicinity of the contour label.

When **Break Contours at Label** is checked, the contour lines will be broken and trimmed at the label location for label visibility.

When **Draw Broken Segments** is checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label By Distance places the labels by distance along the contour. The user is not prompted for screen picks of contour crossing when this option is used.

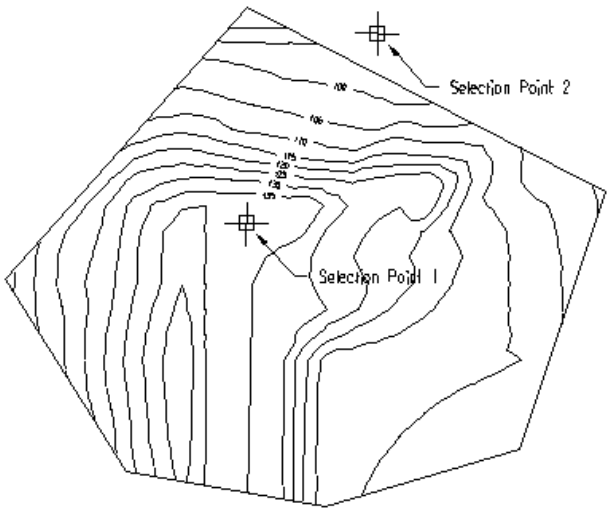
- **Interval** sets the distance interval to be used between labels on each contour.

When **Draw Box Around Text** is checked, a rectangle will be drawn around the elevation labels.

When **Align Text with Contour** is checked, contour elevation labels will be rotated to align with their respective contour lines.

Prompts

Contour Label Options Dialog Opens Select the desired options and press OK.
Define a line which slices the contours at the desired label locations.
Pick 1st point: *pick a point*
Pick 2nd point: *pick a point*



By selecting two points the contour lines that cross the line defined by the two points are labeled.

Keyboard Command: gclabel
Prerequisite: polylines with elevation (contour polylines)

Color Contours by Elevation

Function

This command changes the color of the selectedpolylines based on the polyline elevation. The color to use isdefined in elevation range table.

Prompts

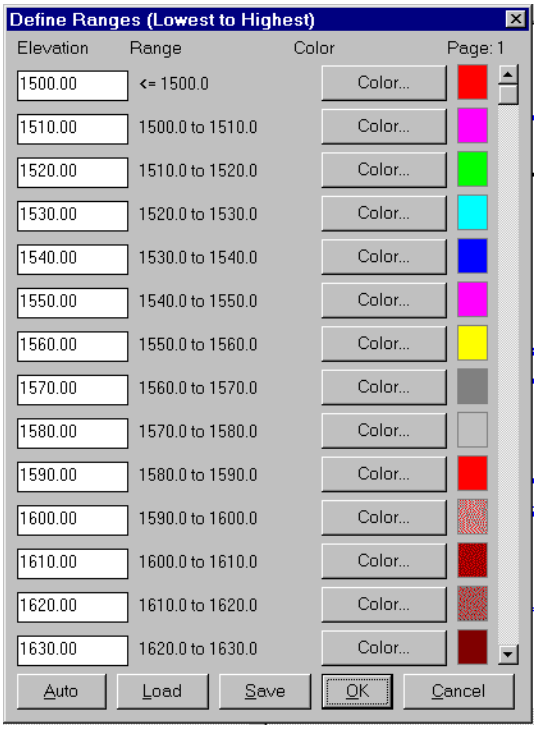
Define Ranges Dialog

Select polylines to color: *pick the polylines*

Pick point for color legend: *pick a point to a clear areaof the drawing to place a legend or press Enter for no legend*

Prerequisite: Contours polylines

Keyboard Command: ctrcolor



Color Contours by Interval

Function

This command changes the color of the selected polylines according to their interval. The polylines can be selected from the screen or chosen by their layer.

Prompts

Command: ctrcolor2

Select polylines to color.
Select objects: *1 found*

Select objects: *1 found, 2 total*

Select objects: *1 found, 3 total*

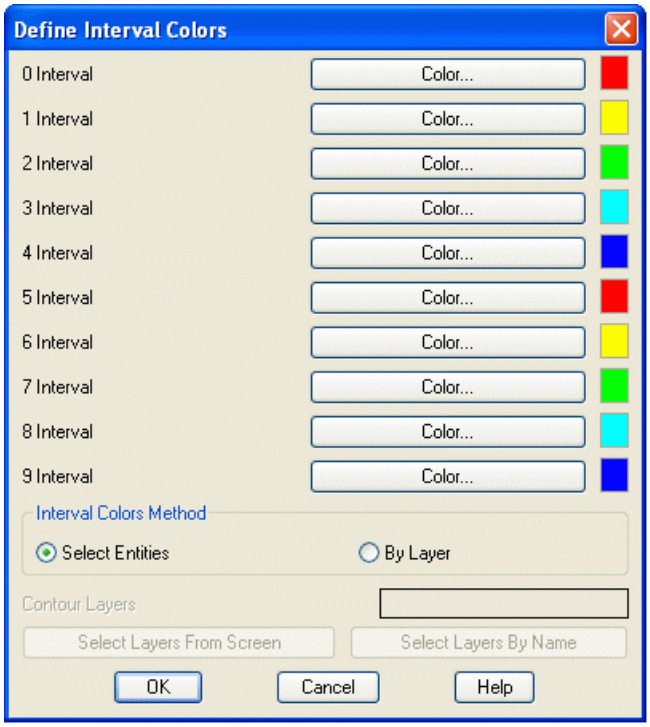
Select objects: *1 found, 4 total*

Select objects: *1 found, 5 total*

Select objects: *Press Enter to end*

Prerequisite: Contours polylines

Keyboard Command: ctrcolor2



Draw Triangular Surface

Function

This command draws the triangulation that is stored in a triangulation (.TIN) file. You havethe option of plotting the triangulation as 3D LINES or 3DFACEs. 3DFACE entities can be shadedusing the AutoCAD shade command or shaded while using 3D Viewer Window or 3D SurfaceFlyOver. 3D Lines cannot be shaded.

Triangulation (.TIN) files can be created by*Triangulate & Contour*.

Prompts

Select TMESH File to Draw

Choose a triangulation (.TIN) file

Layer name <TMESH>: *Press Enter*

Draw TIN as 3D Lines or 3DFaces [Lines/<3DFaces>]?*Press Enter*

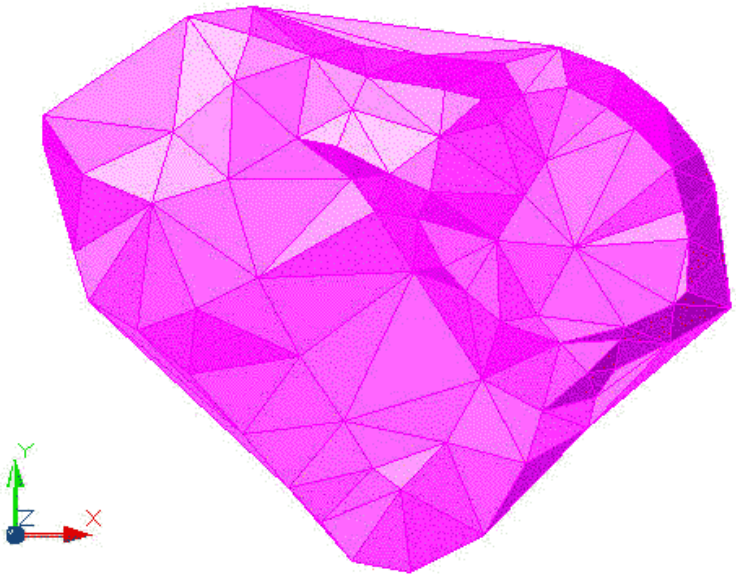
Loading edges...

Loaded 198 points and 234 edges

Prerequisite: A triangulation (.TIN) file

Keyboard Command: drawtri

Before and after of swapping middle triangulation edgB



Triangulation mesh drawn as 3DFaces and shaded with the AutoCAD Shade command

Run Off Tracking

Function

This command draws 3D polylines starting at user picked points downhill until they reach a local minimum or the end of the TIN. In effect it simulates the path of a rain drop. The surface is modeled by a tin file created by 'make existing' or 'make design' surface. The program also reports the horizontal and slope distances, average slope, maximum slope, and vertical drop. These values can be used for time of concentration calculations. Runoff tracking is a convenient way to identify distinct watershed areas.

Prompts

Enter the run off path layer <RUNOFF>: *Press Enter*

tin File Selection dialog box

Choose the triangulation file that models the surface.

Enter rainfall level <6.00>: *Enter*. This allows the runoff line to continue past flat or low points in the grid or TIN, by allowing these area to fill up with water, in essence, up to the specified depth, thus letting the runoff polyline continue on.

Draw tracking for all grid cells or pick individualls [All/<Pick>]: *Enter*. Pressing Enter leads to individual picking of runoff tracking lines, while A for All would fill the drawing with runoff polylines.

Pick origin of rain drop: *pick a point at the top of the run off polyline*

Pick origin of rain drop (Enter to end): *Press Enter*

Pull-Down Menu Location: Draw

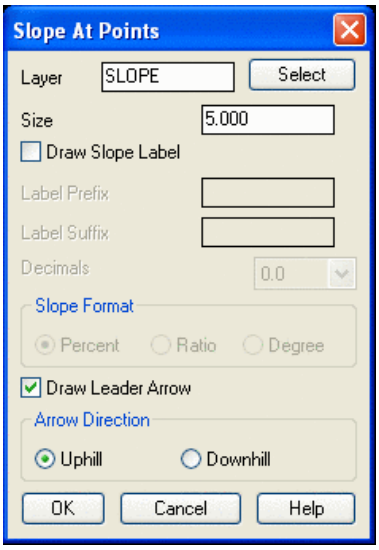
Prerequisite: a tin file

Keyboard Command: runoff

Slope_At_Point

Function

This command allows you to label the slope of a selected surface at a selected point.



In the above dialog you can choose the layer name for the labeling, the size of the label, enter in a label Prefix or Suffix, pick the number of decimals to report the slope out to, and also the direction the arrow is draw,

uphill or downhill.



Prerequisite: a surface

Keyboard Command: ptslope

Cut Fill Map Legend

Function

This command will draw a Cut/Fill Map Legend on your drawing. It will display the cut/fill amount, color, and range, as seen below.

Prompts:

Command:
CF_MAP_LEGEND
Select point for color legend:
Legend size <10.0>:

Prerequisite: Cut/Fill amounts

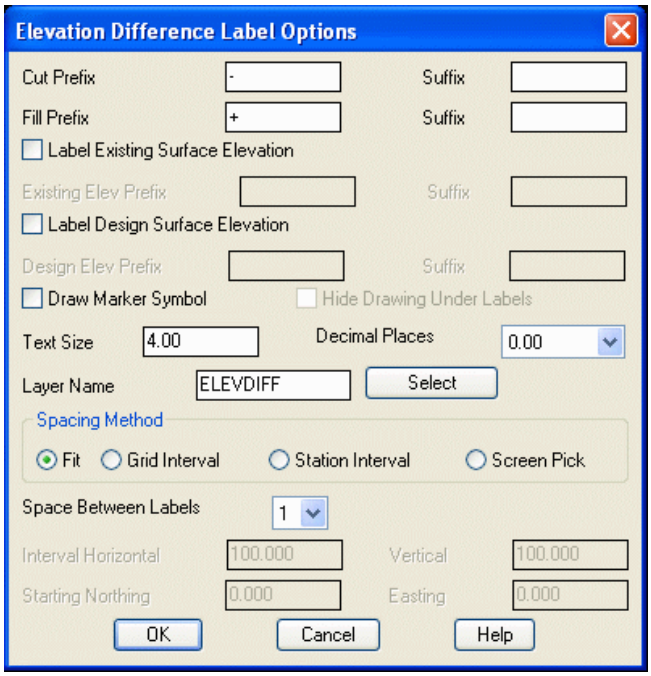
Keyboard Command: CF_MAP_LEGEND

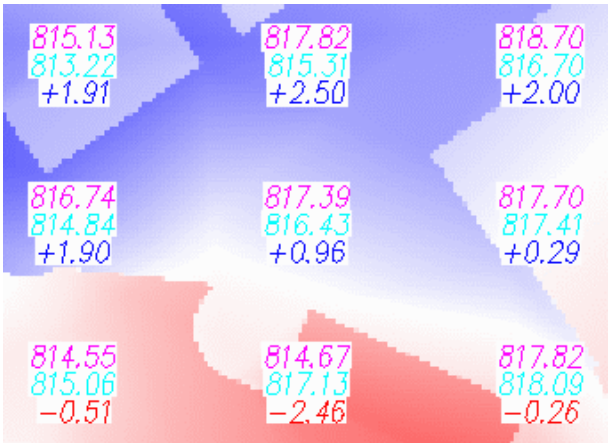
Cut_Fill_Labels

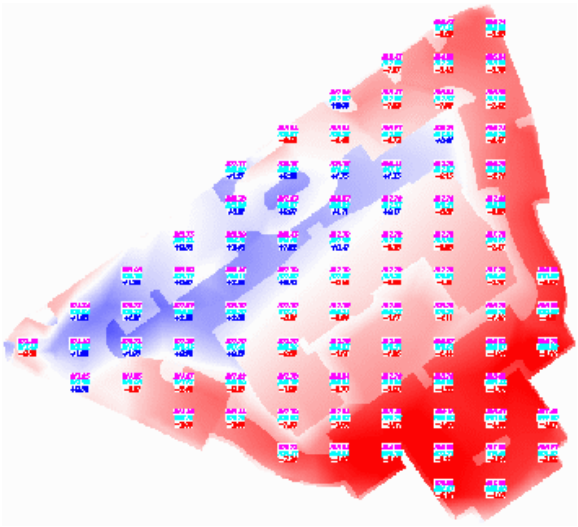
Function

This command displays the design elevation, the existing elevation, and the amount to either cut or fill right on the screen. In the below dialog you can customize the Cut/Fill labels. Text can be added either before or after the

Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations, the Design Surface elevations and Strata Cut Thickness. Carlson TakeOff gives you the option to draw a marker symbol for where each label represents. You can also hide the drawing under the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Decimal Places sets to how many decimal places the labels will report. The Spacing of the labels can be determined by intervals or by a selected number of spaces. The size of each space is determined by the Text Size.







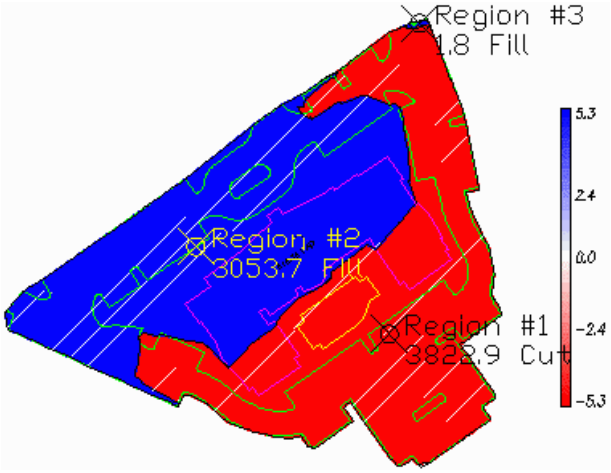
Prerequisite: existing and design surfaces

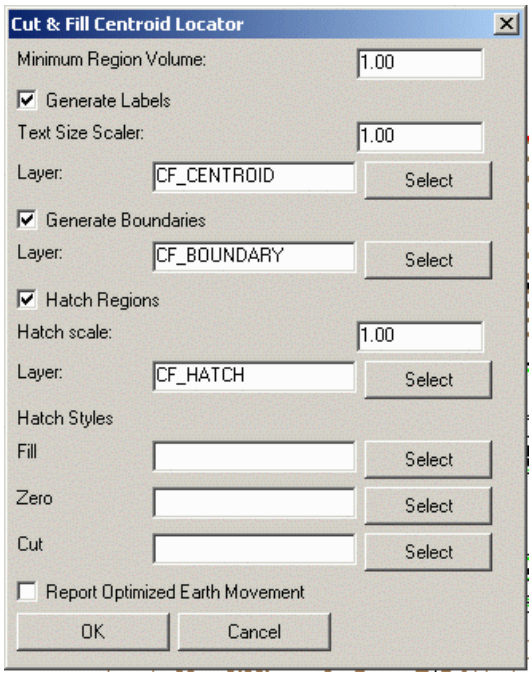
Keyboard Command: elevdiff

Cut Fill Centroids

Function

This command visually shows cut/fill areas in your drawing and finds the center of mass or centriods for every cut and every fill region. Also, a report can be created to show volumes in each region.





Points are created to show the centroids locations. Also, labels can be created to display the volumes in the different regions.

```
Cut & Fill Centroid Report

Original Ground: G:\oem4\src2\work\demo1-ex.flt
Design Surface:  G:\oem4\src2\work\demo1-fn.flt

Region  Volume(C.Y.)     Northing      Easting
1       3822.9    Cut   2190032.48    6135170.29
2       3053.7    Fill  2190117.40    6134982.48
3       1.8       Fill  2190334.67    6135199.41

Earth Movement Report:
From Region  To Region  Volume(C.Y.)  Distance
1            2          3053.7        206.12
1            3          1.8           303.59
1            External   767.4
Total Internal Volume * Distance: 629972.20
Total External Volume: 767.44
```

Here is the Cut and Fill Centriod Report for the above example. It shows the volumes, the coordinates of the centroids, and the Earth Movement Report. The Earth Movement Report shows the minimal distances for moving Cut to Fill areas.

Prompts

Command: tk_cutfillc

Loading edges...
Loaded 1199 points and 3391 edges
Created 2193 triangles

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading edges...
Loaded 8732 points and 25651 edges
Created 16920 triangles

Generating report:
Region #2
3053.7 Fill
Reading the selection set ...
Joining ...
Joined 437 entities.
Region #1
3822.9 Cut
Reading the selection set ...
Joining ...
Joined 636 entities.
Region #3
1.8 Fill
Reading the selection set ...
Joining ...
Joined 49 entities.

Prerequisite: Existing and Design surfaces

Keyboard Command: tk_cutfillc

Point Defaults

Function

This command sets Carlson point options.

Descriptions: Specify whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

Elevations: Specify whether you are prompted for a point elevations when creating points and whether the point elevations are labeled in the point block.

Locate on Real Z Axis: When checked, points are located at their actual elevation, otherwise points will be located zero elevation.

Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts

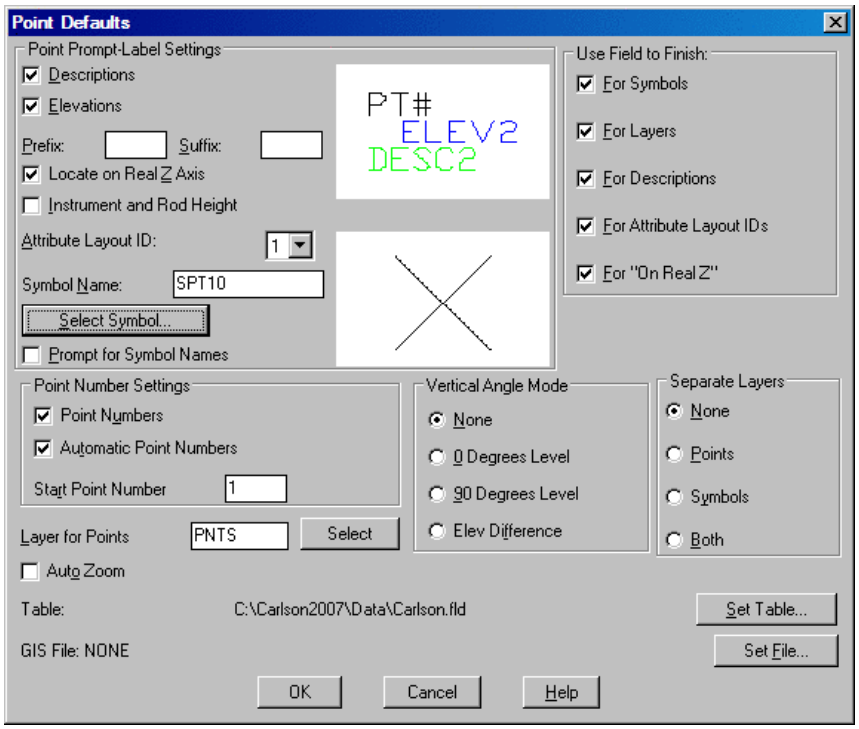
are defined in AutoCAD drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing.

Symbol Name: Enter the default symbol name to use. You may also pick the Select Symbol button to select a symbol from the symbol library.

Prompt for Symbol Names: When checked, you will be prompted for each symbol name instead of using the default symbol.

Point Numbers: When this toggle is OFF, no point number will be created and no points will be stored in the coordinate (.CRD) file.

Automatic Point Numbers: When this toggle is OFF, commands that locate a point will prompt for a point number. Otherwise, point numbers are numbered sequentially. If the Start Point Number field is set to 0, no point will be plotted. An exception to this is when you use the *Draw-Locate Points* command and use the Range option, then a point entity is plotted.



The following table illustrates the effects of elevation settings:

```
          Elevations Yes  Real Z No
Picked Point  Labels point, Prompts for elevation, uses 0 for z coordinate
Point Number  Labels point, No Prompt, uses 0 for z coordinate

          Elevations Yes  Real Z Yes
Picked Point  Labels point, Prompts for elevation for z coordinate
Point Number  Labels point, No Prompt, uses z coordinate from file

          Elevations No   Real Z No
Picked Point  No Label, No Prompt, uses 0 for z coordinate
Point Number  No Label, No Prompt, uses 0 for z coordinate

          Elevations No   Real Z Yes
Picked Point  Labels point, No Prompt, uses z coordinate of picked point
Point Number  Labels point, No Prompt, uses z coordinate from file
```

Start Point Number: Specify the next point number to use.

Vertical Angle Mode: Specify how Carlson should prompt you for vertical angles. None means no prompt. Applies to creating points with commands such as *Traverse.* The vertical angle is used to calculate the point elevation.

Separate Layers: Specify settings for point attribute layers.

None: The point symbol, point number, elevation and description use the layer names PNTMARK, PNTNO, PNTELEV and PNTDESC.

Points: The point number, elevation and description layers are composed by concatenating the point layer and the string NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the attribute layers will be UTILNO, UTILELEV and UTILDESC.

Symbols: The point symbol layer is composed by concatenating the point layer and the string MARK. For example, if the point layer is UTIL then the symbol layer will be UTILMARK.

Both: The point symbol, point number, elevation and description layers are composed by concatenating the point layer and the string MARK, NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the symbol/attribute layers will be UTILMARK, UTILNO, UTILELEV and UTILDESC.

Layer for Points: Specify the layer name for Carlson points.

Auto Zoom: When checked, AutoCAD will perform a Zoom—Center around new points to keep the display centered around current working area. This only applies during commands such as Traverse. This setting is also available in *Configure Carlson* under General Settings where it is called Auto Zoom Center for New Points.

Use Field to Finish For: Allows you to use the code definitions from Field to Finish for the Point Symbols, Layers, Descriptions, Attribute Layout IDs and whether to locate the point on the "Real Z" and whether to Separate Attribute Layers when creating new points. For example, when creating a point with description "EP", Carlson would look up "EP" in the Field to Finish table and will use the field code definitions to establish the point instead of the definitions defined in Point Defaults.

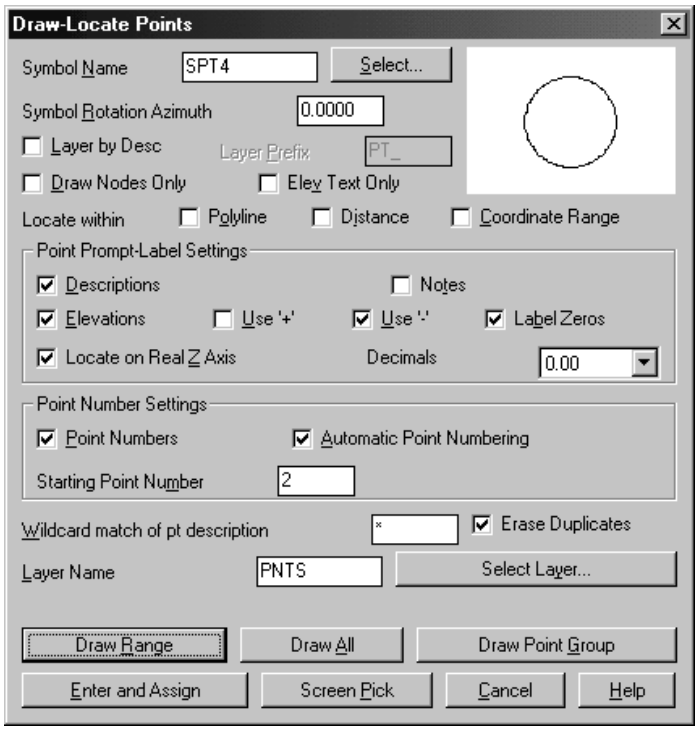
GIS File: This option lets you specify a GIS file to be used when creating new points. The GIS file contains a list of fields to prompt for. For each point that is created, the program will prompt for these fields and store the results to the note file (.not) associated with the current CRD file.

Keyboard Command: ptsetup
Prerequisite: None

Draw-Locate Points

Function

The Draw-Locate Point dialog box allows you to insert either new or existing points into the drawing. You can create new points either by picking points on the screen or by entering northing and easting coordinates. You can place existing points by entering point numbers which reference the current coordinate file. You are prompted to choose a coordinate file if no coordinate file is current.



1 In the Draw-Locate Point dialog box, you must select the point symbol and select placement options if necessary.

- **Symbol Name**: The name of the symbol file is displayed here. You can choose a different symbol by clicking Select. The selected point symbol is displayed on the right.

- **Symbol Rotation Azimuth**: This is the rotation angle that is used for the point symbols. This angle is used in a counterclockwise direction relative to the current twist screen.

- **Layer by Desc**: This option inserts the points in the layer named by the point description. Using Layer by Desc organizes the points by description and allows for layer management. For example, you can use the Isolate Layers command to show only points on a certain layer. If you include an invalid layer character in the description, the layer name stops at the bad character. For example, a point description of "UP / 105" would use layer "UP". The Layer Prefix is added to the beginning of the layer name. For example, a Layer Prefix of "PT_" and a point with the description "EP" would use the layer "PT_EP". Layer Prefix is optional. It allows all the point layers to be grouped.

- **Draw Nodes Only**: This option inserts only a point entity (the node) and not the point block and symbol. This option is most useful when you have a lot of points to insert, because inserting only the nodes is faster than inserting nodes with the point block and symbol. Commands such as Triangulate & Contour and Make 3D Grid File can use these points and do not need the point block and symbol.

- **Elevation Text Only**: Selecting this option draws text of the point elevation without the point block, symbol, or node. The decimal place of elevation text is placed at the northing and easting point location.

- **Locate within Polyline**: This option inserts only the points that are inside a closed polyline. The command prompts you to select a closed polyline. All the points in the current coordinate file are checked. Any points that are located within the closed polyline are drawn.

- **Locate within Distance**: This option inserts only the points that are within a specified distance from a reference point. The command asks you for the reference point and the search distance. All the points in the current coordinate file are checked. Any points that are located within the search distance of the reference point are drawn.

- **Locate within Coordinate Range**: This option inserts only the points that are within the specified range of northing, easting, and elevation. The command prompts for the minimum and maximum northing, easting, and

elevations. These values default to the actual minimum and maximum in the coordinate file. Then the command prompts for the point number range of points to check. The points that fall in both the point number range and the coordinate range are drawn.

2 Under Point Prompt-Label Settings, you determine attributes for which you will be prompted.

- **Descriptions**: This option determines whether you are prompted for descriptions for each point when creating new points. When you are placing both new and existing points, Descriptions determine whether this attribute is labeled with the point inserts.

- **Notes**: This option works with the note file (.not) associated with the current coordinate file. The note file contains unlimited point descriptions in addition to the fixed 32 character point descriptions in the coordinate file. When you create points with Notes on, the command will prompt for point notes to be stored with the point. When you draw existing points with Notes on, any notes for the points are drawn as text entities below the point description.

- **Elevations**: This option determines whether you are prompted for elevations for each point when creating new points. When you are placing both new and existing points, Elevations determine whether this attribute is labeled with the point inserts.

- **Use '+'**: This option labels the positive elevations with a leading '+'. For example, "+159.43".

- **Use '-'**: This option labels the negative elevations with a leading '-'.

- **Locate on Real Z Axis**: This option determines if the points are placed at their elevations or at zero elevation.

- **Label Zeros**: This option will label points with zero elevation when the Elevations option is on. Otherwise only points with nonzero elevation will be labeled.

3 Under Point Number Settings, you determine how points will numbered.

- **Point Numbers**: This option determines whether the complete point block is drawn or just the symbol and node. When you create new points with Point Numbers off, no points are stored in the current coordinate file, and only the point symbol and node are drawn. When you draw existing points with Point Numbers off, the point attribute block is not drawn and only the point symbol and node are drawn.

- **Automatic Point Numbering**: This option applies to creating new points. With this option active, the command will use the **Starting Point Number** for the first new point. The next point number is automatically incremented. Before storing the point, the command checks whether the point number is used. If the point number is used and point protect is on (set in the Coordinate File Utilities command), then the command will prompt for another point number or to overwrite the point. With Automatic Point Numbering off, the command will prompt for the point numbers.

4 Determine how the points are to be displayed and in what layer.

- **Wildcard match of point description**: You can display only points with specific descriptions. This can be thought of as a filter. For example, entering IP, would display only points that are labeled with the description IP, or Iron Pin. The default is the asterisk (*), this will display all points regardless of description.

- **Layer Name**: This allows you to designate a layer for the points to be displayed. You can enter a new name or choose an existing layer by clicking **Select Layer**. An Carlson point consists of a block insert with attributes, a point symbol, and a point entity. The point entity is used for picking the point by OSNAP Node in other commands. The block insert includes a point number, elevation, and description. These attributes are in the PNTMARK, PNTNO, PNTELEV, and PNTDESC layers. The points are also in an overall layer as specified in this dialog box. This layer setup allows you to freeze a group of points by the main layer name or freeze point attributes for all the points in the drawing. For example, freezing layer "PNTS" would freeze all the points in this layer. Freezing layer "PNTELEV" would freeze the point elevation attribute for all the points.

- **Draw Range**: Thiscommand will draw existing points from the current coordinate file. The Draw Range button will prompt for the point numbers to draw.

- **Draw All**: This command will draw all the points in the coordinate file and then zoom the extents of the display to show the points.
- **Enter & Assign**: This command can be used to create new points using the point northing and easting.
- **Screen Pick Point**: This command allows you to create points by picking the point coordinate on the screen. For example, you could set the Object Snap to EndPoint and pick the end point of a building polyline to create a point at the building corner.

Prompts

To create a new point.

In the Draw-Locate Point dialog box Choose Screen Pick Point.

Pick point to create: **pick point**

Select/<Enter Point Elevation <0.00>: **Enter elevation or Press S and enter to select text to set elevation.**

Enter/<Select text of elevation>: **Select text entity that defines elevation of point.**

These prompts appear only if elevation prompting is turned on.

Enter point description: **HUB**

This prompt only appears if description prompting is turned on.

To locate a point in the coordinate file (point number 3 in this example).

1 In the Draw-Locate Point dialog box Choose Draw Range.

Point numbers to draw: **3**

PtNo. North(y) East(x) Elev(z) Description

3 4154.28 4147.35 0.00

2 Point numbers to draw: **1-2**

Locates a range of points. From 1 to 2.

PtNo. North(y) East(x) Elev(z) Description

1 4252.76 4158.32 0.00 RADPT

2 4258.11 4059.38 0.00

3 Point numbers to draw: **Press Enter**

This ends the command.

Prerequisite: You may want to execute the Drawing Setup command under the Setting menu to set the scale and size. You also need a CRD file.

Keyboard Command: LPOINT

Field to Finish

Function

This command turns data collector field notes into a final AutoCAD drawing by matching the descriptions of the field points with user-defined codes. The points are brought into the drawing with attributes defined by the code including layer, symbol, size, line type, etc. Field to Finish replaces PointCAD. Field to Finish uses an improved

coding method but still handles the PointCAD method which should make switching to Field to Finish easier for PointCAD users.

Two files are used in Field to Finish - a data file and a code file. The data file consists of x,y,z points with text description fields. The description fields contain codes for the Field to Finish processing. The data file can be either a coordinate (.CRD) file or an ASCII file. The code file defines the layer, symbol, size and other actions to apply with each code. These file names are displayed at the top line of the Field to Finish dialog box.

Field to Finish can translate the field points into Carlson TakeOff points with a symbol, layer, and size defined by the code. The point settings of whether to label the description, point number, and elevation and whether to locate the point at zero or at the real Z are defined in the Point Defaults routine under the Points menu. The Draw-Locate Points command provides a simpler method for drawing points compared with Field to Finish.

There are two different methods for connecting linework. One method creates line work by connecting points with the same code. The line type is defined by the code as either points only (no line work), lines, 2D polylines, or 3D polylines (breaklines). Distinct lines with the same code are defined by adding a group number to the end of the code name in the data file. With this method, all points with the description CODE1 will be one line while points with CODE2 will be another line. Both CODE1 and CODE2 use the definition for CODE. For example, the code EP could be a code for edge of pavement that is to be connected as 3D polylines. If there are two separate edge of pavement lines on the left and right sides of a road, all the points for the left side could have the description EP1 and the points on the right side could be EP2.

The second method is the PointCAD format. This method also connects points with the same code. The difference is that instead of using a number after the code for distinct lines, you use the same code with an additional code for starting and ending the line. For example, +0 is used to start a line and -0 to end. So the coding for a segment of edge of pavement could be EP+0, EP, EP, EP-0. Another special code that has been added to Field to Finish is +7, -7. This 7 code will use the line type definition of line, 2D polyline or 3D polyline defined by the Field to Finish code. For example, if EP is defined as a 3D polyline, then the coding EP+7, EP, EP, EP-7 will create a 3D polyline. Otherwise codes like +0, -0, which is defined as start and end line, will draw EP as a line.

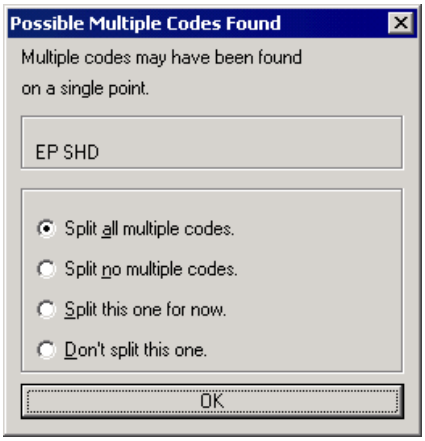
The advantage to the PointCAD method is that you don't have to keep track of line numbers. For example, if you are surveying 50 curb lines, the first method would require you to use 50 distinct curb numbers. The advantage to the first method is that you don't have to use the start and end codes. Also the Nearest Found connection option applies to the first method.

The main Field to Finish dialog box allows you to load the data and code files, view and edit the code definitions, and then process the files. The top section displays the code definitions. The middle section has two rows of buttons for changing the selected code definitions. The bottom section has three columns of functions.

Carlson TakeOff points in the drawing have point attributes including a description. When Field-to-Finish draws the points, the point description from the data file is processed to match a code. The code then defines the description that is drawn with the point. For example, consider a code of "UP" with a description of "POLE" and a data point with the description "UP". The data point description "UP" would be matched with the code "UP" and the point would end up being drawn with the description "POLE". A special character "/" (the divide key) can be used for an unprocessed description to append. Everything after the "/" is added directly to the point description and is not considered a code. For example, a data point with the description "UP / 150" with the same code "UP" definition above would be drawn with the description "POLE 150".

Multiple codes are defined by including each code in the point description field separated by a space. A single data point can be used in different lines by assigning it multiple codes. For instance, a point might be part of both a curb line and a driveway line with a description of "CURB DRW". Field-to-Finish uses spaces as the delimiter for multiple codes. You should avoid spaces in the descriptions except for where multiple codes are intended or after the "/" character. For example, a code for light post should not be "LGT POST" but instead could be "LGTPOST". When Field-to-Finish detects spaces in the descriptions at start up, there is an option of whether to process the multiple

codes.



Field to Finish recognizes several special codes suffixes. A special code comes after the regular code. A space separates the codes. Here is a listing of the default special codes.

Special Codes

PC

This code begins a three point arc. The point with this special code is the first point on the arc. The next point with the code is considered a point on the arc, and third point with the code is the arc endpoint. For example (in point number, X, Y, Z, description format),

10, 500, 500, 0, EP PC - start curve

11, 525, 527, 0, EP- second point on curve

12, 531, 533, 0, EP- end point of curve

PT

This is a special code that can be used with "PC" to define a curve with more than three points. Starting at the point with the "PC", the program will look for a "PT". If the "PT" is found, all the points between the "PC" and "PT" are used for the curve which is drawn as a smoothed polyline that passes through all points and only curves the polyline between points. If no "PT" is found, then the regular three point arc is applied as explained above.

CLO

This code forces the lines drawn between a series of points with the same code to close back to the first point with the same code. For example, shots 1-4 all have the BLD description with the exception of point 4. Its description is BLD CLO. This will force the linework drawn for the BLD code to close back to point 1 which is the first point with the description of BLD.

NE

This code represents no elevation. A point with this special code is located at zero elevation.

OH & OV

The codes "OH" and "OV" stand for offset horizontal and offset vertical. These offset codes apply to 2D and 3D polylines. A single set of offset codes can be used to offset the polyline a set amount. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP

12, 531, 533, 103, EP

This would create a polyline connecting points 10,11 and 12 and an offset polyline with a 2.5 horizontal and -0.5 vertical offset. The direction of the horizontal offset is determined by the direction of the polyline. A positive horizontal offset goes right from the polyline direction and a negative goes left. The horizontal and vertical offset amounts apply starting at the point with the offset codes until a new offset code or the end of the polyline. Only one horizontal and vertical offset can be applied to 2D polylines. For 3D polylines, multiple offset codes can be used to make a variable offset. For example,

10, 500, 500, 100, EP OH2.5 OV-.5

11, 525, 527, 101, EP OH5.5 OV-.75

12, 531, 533, 103, EP OH7.5

This would offset the first point horizontal 2.5 and vertical -0.5, the second point horizontal 5.5 and vertical -0.75 and the third point horizontal 7.5 and vertical -0.75.

SZ

This code is used to set a different symbol size. The value of the new symbol size is specified after the SZ (example SZ0.2). This value is a size scaler that is multiplied by the current drawing scale to determine the actual drawn size. For example, a drawing scale of 50 and a symbol size scaler of 0.2 would make the drawn symbol size 10.

ROT

This code is used to set the rotation of the point symbol. If a point number follows the ROT code, then angle from the current point to this point number is used for the rotation. For example, "ROT45" would rotate the symbol towards point number 45. If there is no point number after the ROT code, then the rotation point is the next point number with the same code as the current point.

SMO

This code is used to smooth the polyline.

SCA

This code is used to control multi-point symbols described later in this section.

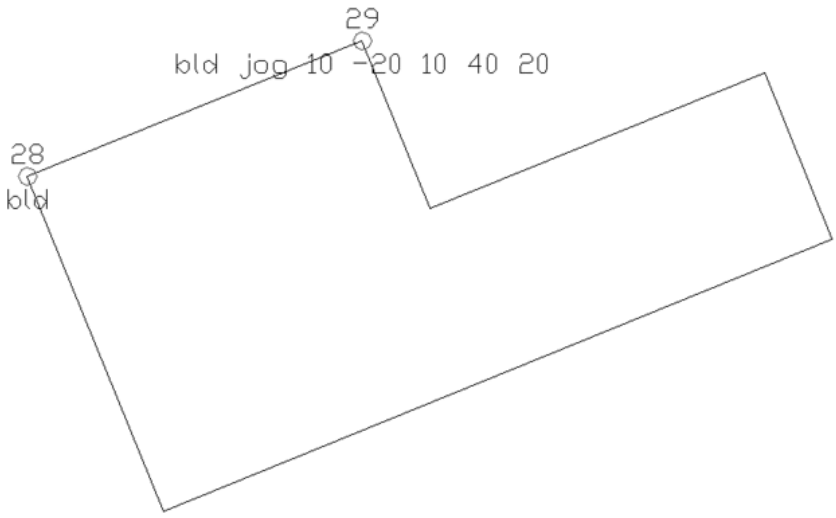
AZI & DIST

The AZI and DIST codes are used together to locate an offset point. The AZI sets the offset azimuth and DIST sets the distance. The values should directly follow the code. For example, AZI25 DIST4.2 would draw the point offset 4.2 at an azimuth of 25 degrees.

JOG Special Code

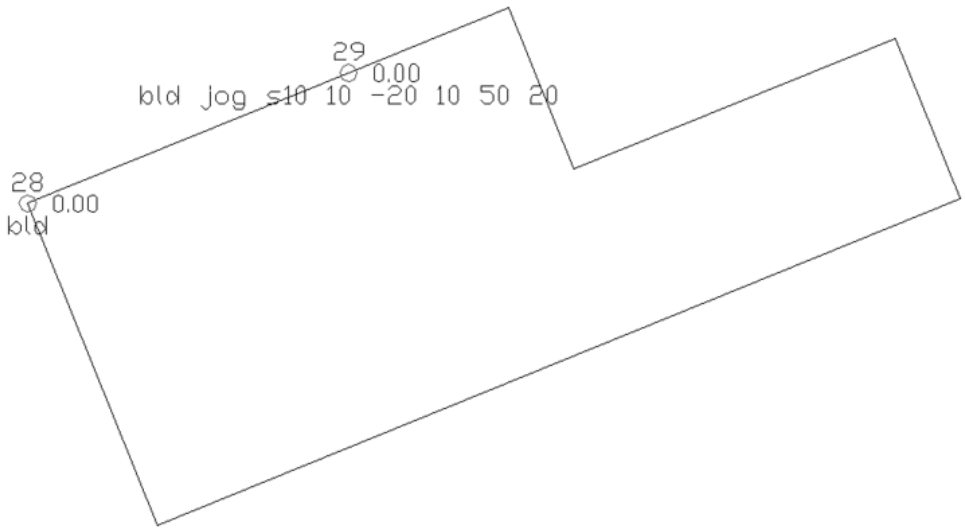
The "JOG" special code allows for additional points to be inserted into the line work at perpendicular offsets. Only offsets should follow the JOG code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. Alternatively, "R#" and "L#" can be used where # is the distance to either the right or the left. For example,

"BLDG JOG R5 L12.2 L5 L12.2" or equivalently "BLDG JOG 5 -12.2 -5 -12.2" draws a closed rectangle on the right hand side of an existing line. The offsets are always done in the X-Y plane. If the current line is vertical, an offset to the right is along the positive X-axis. Just as "cb pc" shown above uses the "pc" special code to launch into a 3-point arc, by default, the "jog" special code, following any normal user-defined code, enables the entry of left and right "jogs" or segments of a polyline. This is useful for drawing buildings based on tape measurements, as opposed to trying to physically survey each building corner by total station or GPS. The key is to take 2 measurements on a building, which establishes a "line" or vector. Assuming you used bld for building, the second bld would be followed by the reserved word "jog", which in turn would be followed by the left and right measurements in the form of bld jog 10 -20 10 40 20. The "-" or negative sign indicates a left-hand jog. All "jogs" are at right angles to the previous segment. With Carlson TakeOff, the JOG option no longer creates duplicate points for each jog segment. The additional segments are drawn with no associated point numbers, minimizing point file size.



Straight JOG within JOG command

It is possible to add a straight "jog" instead of the conventional right and left jogs. This is done by using the S# option, as in S10, for 10 units, which must follow the "JOG" expression. You cannot use S10 as a reserved, "understood" command unless it follows JOG. This "straight jog" option makes the most sense when there is a need to extend the vector or line of the first 2 building points surveyed. If you can't see or obtain a shot at the true, second building corner, you could take 2 shots where you can obtain them, then measure the additional distance to the true corner, recording it as a straight jog. Here is an example, displayed graphically.



JPN

The "JPN" (Join to Point Name) special code joins to the point named immediately after the code. For example, "JPN205" causes a line to be drawn from the current point to the point "205".

RECT

The "RECT" special code causes a rectangle to be formed on a 2D or 3D polyline using one of two different methods. If a number follows "RECT" (e.g., "RECT10"), a rectangle will be drawn 10 units to the right of the last two points ending on the point with the "RECT" code. Use a negative offset to place the rectangle on the left side (e.g., "RECT-2.5"). If no number follows "RECT", then the polyline will be closed by shooting right angles from the first point of the polyline and the current point and creating a new point where those two lines cross.

COAL

This code creates a coal section for use in the Mining module.

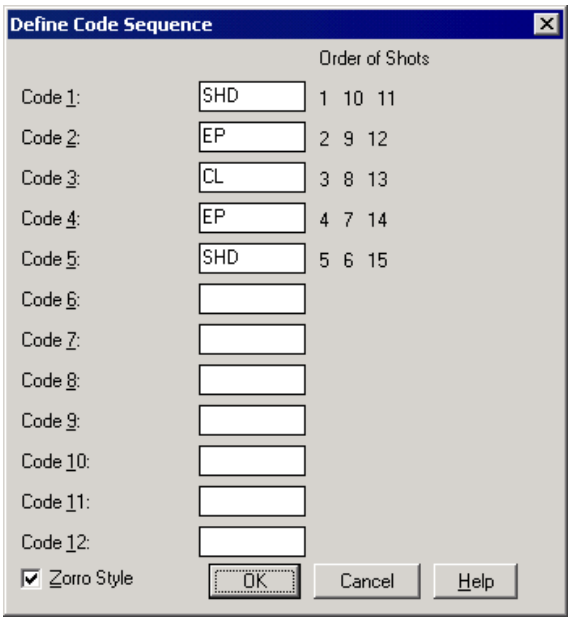
You may substitute your own code for any of these special codes in the Code Definition Settings dialog shown on page 260.

Field-to-Finish will layerize the points and linework according to the code definitions. If the layers to use are not already defined, Field-to-Finish will create the necessary layers and assign different colors. To have the same colors for these layers in all your drawings, define the layers in the prototype drawing. The prototype drawing is the default drawing that is loaded whenever a new drawing is created. To define layers in the prototype drawing, save your current drawing and then start a new drawing with the New command. Don't give the new drawing a name, just click OK. Then define the layers as desired with the Layer command. When you are done creating layers, use the SAVEAS command and change to Drawing Template (.DWT) under Save as Type. In Carlson TakeOff, the default drawing template that is used is named 15SCDRAW.DWT (or 14SCDRAW for AutoCAD R14 users). You can overwrite this default template or make a new drawing template. If you make a new one, you may want to edit the Carlson TakeOff icon to use the new one. To edit the icon, highlight the icon with one click and then click the right mouse button. Choose Properties and then Shortcut and change the drawing template name.

Sequences

Sequences are a way to simplify field entry of a sequence of codes. For example, a road cross-section could be SHD1 EP1 CL EP2 SHD2. Instead of entering these different descriptions, one sequence definition can store these descriptions in order. Then just the sequence code (such as RD) is used in the field. The cross-section can be shot in left to right then left right order, right to left then right to left order, or alternating left to right then right to left order. The alternating method is known as the Zorro style. The one restriction is that the shots always start from a right or left edge.

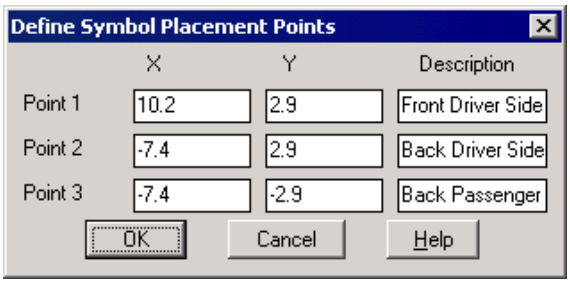
To set up a sequence, choose the Sequence toggle in the Edit Code dialog. Then pick the Define Code Sequence button. This brings up a dialog for entering the sequence codes in order. These sequence codes should be defined as normal codes somewhere else in the Field to Finish code table (ie SHD as a 3D polyline).



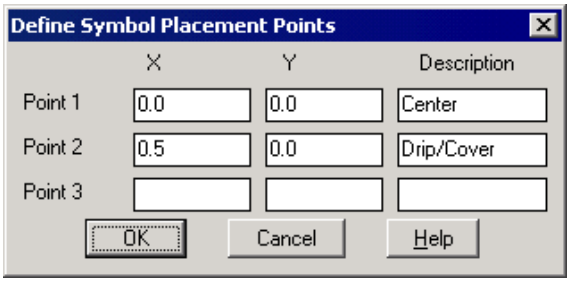
In the field, the one template code is used for all the cross-sections shots (ie RD for all the points). Then Field to Finish will substitute this template code with the sequence codes (ie substitute RD with SHD).

Symbol Points

For each code definition, the symbol insertion points can be defined with up to three points. To define the symbol insertion points, choose the Symbol Pts button in the Edit Code Definition dialog box. By default, the symbol insertion is defined by one point at the symbol center (0,0). A one-point insertion definition can be used to insert a symbol offset from the center. With a two insertion point definitions, the program will rotate and scale the symbol. For example, two insertion points can be used to insert a tree symbol to size the tree, where the first point is for the tree center and the second is for the drip line. With three insertion point definitions, the program will rotate and scale the symbol in both X and Y. For example, three points can be used to insert a car symbol with the first point being the front drivers side, the second point as the back driver side (to rotate and scale the length) and the third as the back passenger side (to scale the width). Besides the insertion point coordinates, you can define a description for each point which is used for the drawn point description and is used for prompting in the Insert Multi-Point Symbol command and in Carlson Software's Tsunami data collection. See a three point symbol example below.

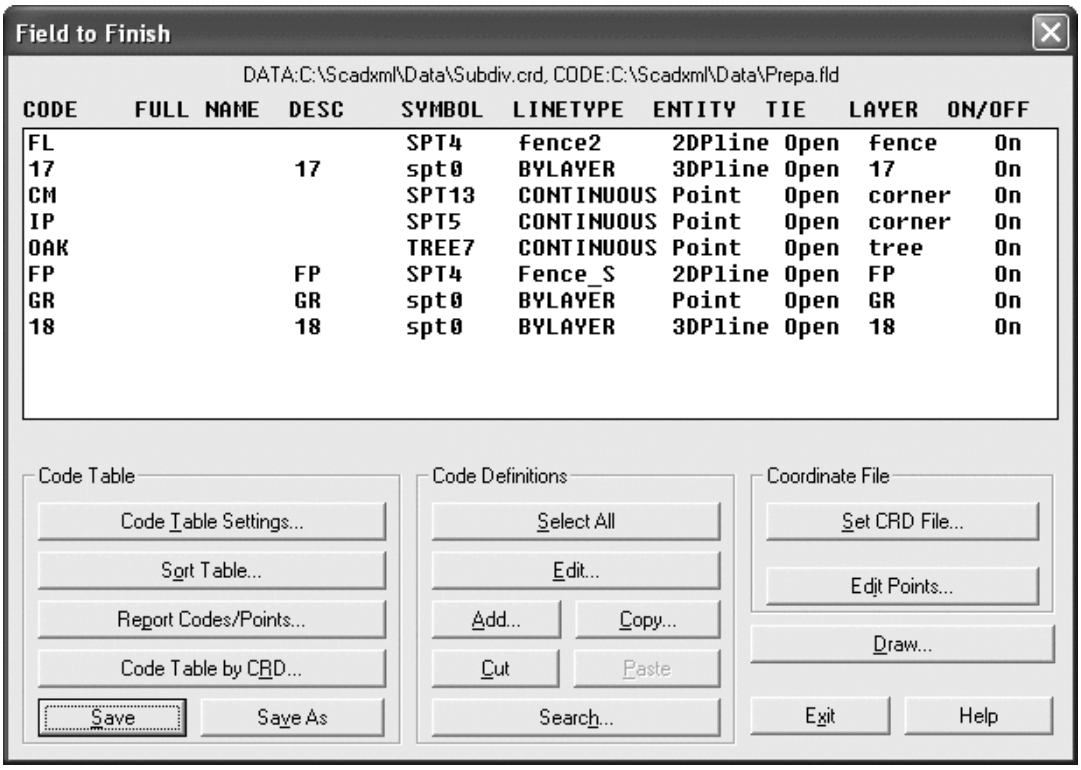


The coordinates for the insertion point definitions are for the symbol at unit size. To figure these coordinates, you will need to open the symbol drawing (.DWG) file. By default, the symbols are located in the Carlson TakeOff SUP directory. For example to make an insertion point for the tree drip line, open the tree symbol drawing and find the coordinate at the edge of the tree symbol (in this case 0.5,0.0). Shown below is a two point symbol example.



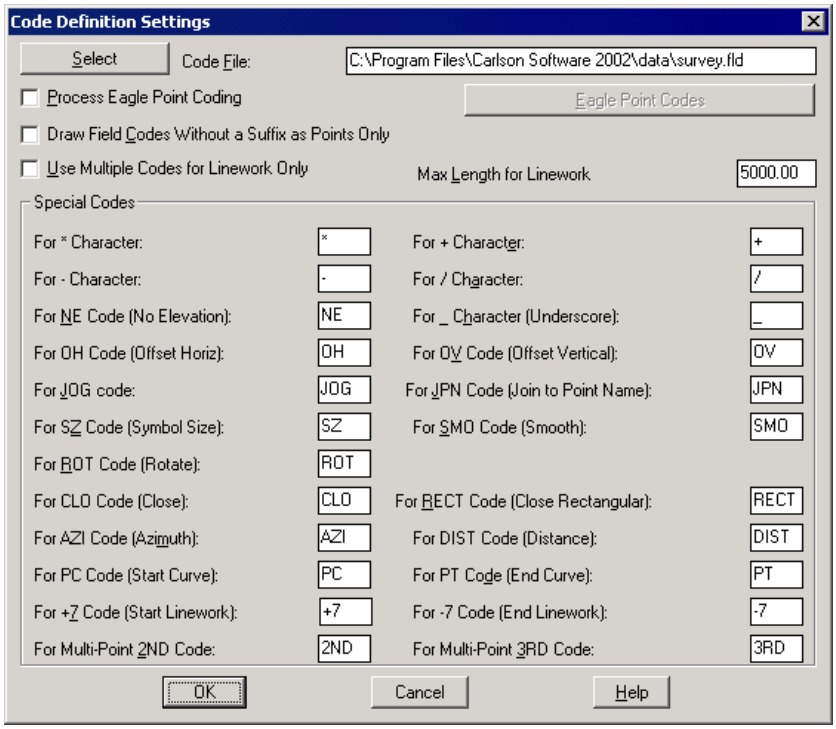
Not all of the symbol insertion points need to be used when drawing the points. If a code definition has a three insertion points, it is possible to use just the first two or first one. There are special codes to associate multiple points to the same symbol. The first code point is used as the first symbol insertion point. The "2ND" code is used to specify the second symbol insertion point. A point number can follow the "2ND" to identify a specific point. Otherwise without the point number, the program will use the next point with the current code. The "3RD" code is used to specify the third symbol insertion point and similar to the "2ND" code, a point number after the "3RD" is optional. The "2ND" and "3RD" codes should be assigned to the first point. For example, consider a code of "CAR" with a three point symbol insertion definition. If point #1 has a description of "CAR 2ND 3RD", then point #1 will be used as the first symbol insertion point and the next two points with the "CAR" description will be used as the second and third symbol insertion points.

Using the Main Field to Finish Dialog



Code Table Settings

1 **Select Code Table** - This command opens the Code Definition Settings dialog shown below.


Code Definition Settings
Select
Code File:
C:\Program Files\Carlson Software 2002\data\survey.fld
Process Eagle Point Coding
Eagle Point Codes
Draw Field Codes Without a Suffix as Points Only
Use Multiple Codes for Linework Only
Max Length for Linework
5000.00
Special Codes
For * Character: *
For + Character: +
For - Character: -
For / Character: /
For NE Code (No Elevation): NE
For _ Character (Underscore): _
For OH Code (Offset Horiz): OH
For OV Code (Offset Vertical): OV
For JOG code: JOG
For JPN Code (Join to Point Name): JPN
For SZ Code (Symbol Size): SZ
For SMO Code (Smooth): SMO
For ROT Code (Rotate): ROT
For CLO Code (Close): CLO
For RECT Code (Close Rectangular): RECT
For AZI Code (Azimuth): AZI
For DIST Code (Distance): DIST
For PC Code (Start Curve): PC
For PT Code (End Curve): PT
For +7 Code (Start Linework): +7
For -7 Code (End Linework): -7
For Multi-Point 2ND Code: 2ND
For Multi-Point 3RD Code: 3RD
OK
Cancel
Help

- **Select**: Choose this button to specify a new code table. The name of the current table is shown in the field to the right of this button.

- **Process Eagle Point Coding**: When checked, this option allows the user to switch from interpreting coordinate files based on the Carlson Field-to-Finish method to interpreting coordinate files where the field coding was entered using the Eagle Point Data Collection method.

Currently the supported designators include, "Line", "Curve", "Close Line", "Stop Line", "Insert Description" and "Bearing Close". Also supported is the ability to recognize overwriting of descriptions just as Eagle Point does by using the space separator instead of the "Insert Description" designator. Examples of supported coding are as follows:

.TC Places a node and or line per the field code library.

TC Places a node and or line per the field code library.

.TC Specifies a point on a curve.

TC- Specifies a point on a curve.

..TC Stops the line.

TC! Stops the line.

.TC+ Closes the line back to the starting point.

TC+ Closes the line back to the starting point.

.TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.

WV.W1 Places a node as specified by the code "WV" in the field code library and then begins a line as specified by code "W" in the field code library.

.TC.EP.FL Results in three lines coming together.

TC1.TC2.TC3 Results in three lines coming together. All three lines are specified by the definition of the single code "TC" in the field code library.

TC.TC1 When used in conjunction with the "Draw Field Codes Without a Suffix as Points Only" toggle, "TC" will be recognized as the node and "TC1" will be recognized as the line so that if the code "TC" in the field code library is defined as a polyline, line or 3D polyline, duplicate lines will not be unintentionally placed when this shot only pertains to a single element. Keep in mind that all line work must have a numeric suffix when using this toggle.

TREE * OAK Result on screen would be: TREE OAK

TREE OAK * Result on screen would be: OAK TREE

TREE OAK Result on screen would be: OAK

TC1!.TC2-.VLT6# Stops "TC1", continues "TC2" as a point on a curve and closes VLT6 as a rectangle using the "Bearing Close" code.

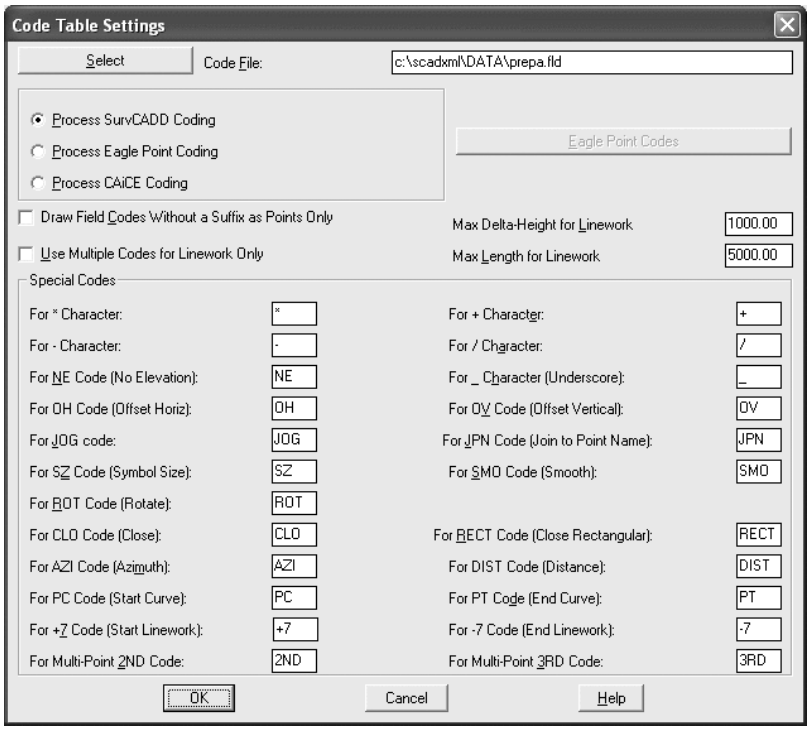
Note: The use of the "Use Multiple Codes for Linework Only" toggle is recommended when using Eagle Point Coding.

- **Use Multiple Codes for Linework Only**: When checked, and multiple codes are detected, only linework will be drawn for the secondary codes. Points are only created based on the primary code. If you want symbols for all multiple codes, then uncheck this setting.

- **Max Length for Linework**: Specify the maximum length that Field to finish should draw any section of linework.

- **Max Elevation Difference for Linework:** With Carlson TakeOff, within the option "Code Table Settings", you can limit the elevation difference within which linework will connect. Thus, if you were taking fence line shots on a ridgeline fence, then took a series of fence line shots in the valley, the ridge linework would stop and not connect

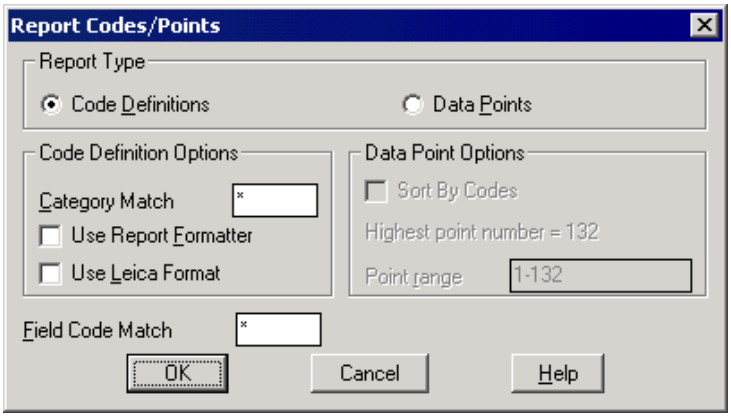
to the valley linework if the elevation exceeded an entered amount. If you wish to disable this "elevation detection" for linework, then keep the setting for the elevation high, as shown below in the upper right of the dialog.



• **Special Codes**: This section allows you to specify your own code for commands such as start curve, end curve, and offsets.

2 **Sort Table** - This sorts the code table by either code name or layer.

3 **Report Codes/Points** - This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.

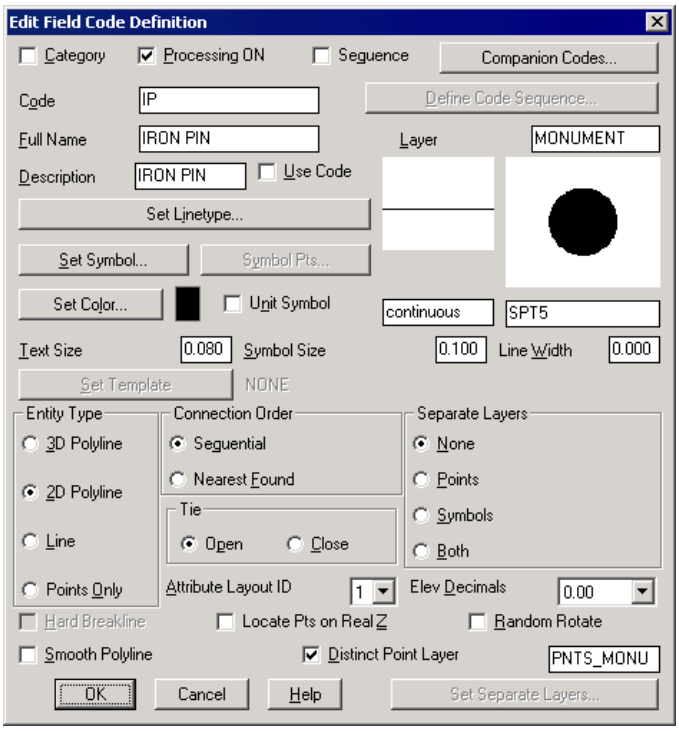


4 **Code Table by CRD** - This command will create code table definitions based on the data file field descriptions. This is useful when creating a code table from scratch.

Code Definitions

1 **Edit** - This command opens the Edit Field Code Definition dialog box. The currently highlighted code will be

edited.



- **Category**: This is an optional field that can to used to help organize your codes. A category is not used for processing and only is useful in viewing and printing.
- **Sequence**: This specifies a sequence type code. Sequences are described above in this section.
- **Define Code Sequence**: This sets the code names that make up the sequence.
- **Processing ON**: This toggle controls whether this code will be processed.
- **Code Name**: This is the key name that identities the code and is matched with the field data descriptions.
- **Layer**: The point and line work for the code will be created in this layer.
- **Full Name**: This is an optional field that describes the code for viewing.
- **Description**: This value is assigned to the point description field. An additional description can be added to a point by entering it after a forward slash in the data description field.
- **Use Code**: This option turns off the Description field described above. Instead the points will be drawn with their original unprocessed descriptions.
- **Linetype**: Line work can be drawn in any of the special linetypes or with the linetype for the layer ("BYLAYER"). The spacing and size of the special line types is determined by the AutoCAD LTSCALE system variable and by the line type settings from the Annotate Defaults command. The special line type "hedge" is drawn with a user specified width. The special line type "userdash" is drawn with user specified distances for the length of the dash and the length of the gap between dashes. You will be prompted for this information when you select that line type.

Carlson TakeOff also offers continuous linetypes to Field-to-Finish. When you select "Linetype" or "Set Linetype" in the Edit options, you can choose among the individual entity linetypes (first group of selections) and the true "continuous" linetypes (second part of the selections). The difference in these linetypes is illustrated by the "Copy" command. A fence line made up of individual entities, when copied by a single selection pick, will copy only the single entity picked (eg. the polyline but not the "X's"). A continuous linetype will copy or offset as one entity,

including the "X's" in the fence. This is shown in the graphic below, where the lower fence line is made up of individual entities and upper fence line is a continuous linetype.

- **Symbol**: This is the point symbol for the code. To avoid drawing a symbol, use the Carlson TakeOff symbol named SPT0.
- **Color**: The line work will be drawn in this color. The default is BYLAYER.
- **Symbol Size**: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size in AutoCAD. The horizontal scale can be set in Drawing Setup.
- **Text Size**: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size.
- **Unit Symbol**: This option will draw the point symbol at unit (1:1) scale. For example, this option could be used for a symbol that is already drawn to actual dimensions such as a car symbol.
- **Set Template**: For 3D polyline codes, this option allows you to assign a template (.TPL) file to the code. The code points act as the centerline for the template and the program will drawn parallel 3D polylines for each break point in the template. The template file can be created in Carlson TakeOff.
- **Entity Type**: This defines the line type to be created. Points only does not create any line work. 3D Polyline can be used for breaklines.
- **Hard Breakline**: This will tag the 3D polylines created with this code as hard breaklines. In Triangulate & Contour, contours are not smoothed as they cross hard barriers.
- **Separate Layers**: This controls the layers of the point and symbol attributes. With "None" the point layers are the standard layers, "PNTNO", "PNTELEV" and "PNTDESC", and the symbol layer is "PNTMARK". With "Points" or "Both" the point attribute layers begin with the layer for the code followed by the attribute type. For example, the "DWL" code shown in this dialog has a layer name "DRIVEWAY". The point attributes would then be "DRIVEWAYNO", "DRIVEWAYELEV" and "DRIVEWAYDESC". With "Symbols" or "Both" the symbol attribute layer begins with the layer for the code followed by "MARK".
- **Smooth Polyline**: This applies a modified bezier smoothing to the polyline. The smoothed polyline will pass through all the original points.
- **Connection Order**: The points of a distinct code can be connected in their point number order or by nearest found which makes the line by adding the next closest point.
- **Tie**: When checked the linework drawn with this code will always close. For example if you have points 1, 2, 3, and 4 with the code BLDG and Tie is checked on for the code BLDG, then the linework will be drawn from point 1 to 2 to 3 to 4 and then back to point 1, closing the figure
- **Precision**: This controls the display precision for the elevation label.
- **Attribute Layout ID**: Controls the location of the point number, elevation and description. These attribute layouts are defined in AutoCAD drawings that are stored in the Carlson TakeOff SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing. This option allows you to assign a different point display "style" (referred to as the attribute layout ID) with each particular code. This option is also available when multiple codes are selected. The option appears in the "Multiple Set" dialog. By selecting different attribute "ID's", you can set the location of the point number, elevation and description with respect to the node of the point, change the rotation or set the font and color of these attributes. New "ID Layouts" are made by loading the file for "Srvpno1.dwg" or "Srvpno2.dwg" or any of the "Srvpnox.dwg" files found in the "SUP" subdirectory. The attributes and their colors appear as shown below, and can be edited and re-saved as a new "Attribute Layout ID". To save as ID 6, for example, use SaveAs and save the file as Srvpno6.dwg.
- **Locate Pts on Real Z Axis**: This option will draw the points at the actual point elevation. Otherwise the points are drawn at zero elevation. For example, you could turn this option off for the FH for fire hydrant code to drawn

them at zero. Then the GND code could have this option on to draw the ground shots at their elevations.

- **Random Rotate**: This option will randomly rotate the symbol. For example, this option could be used for tree symbols to have the trees drawn in various orientations.

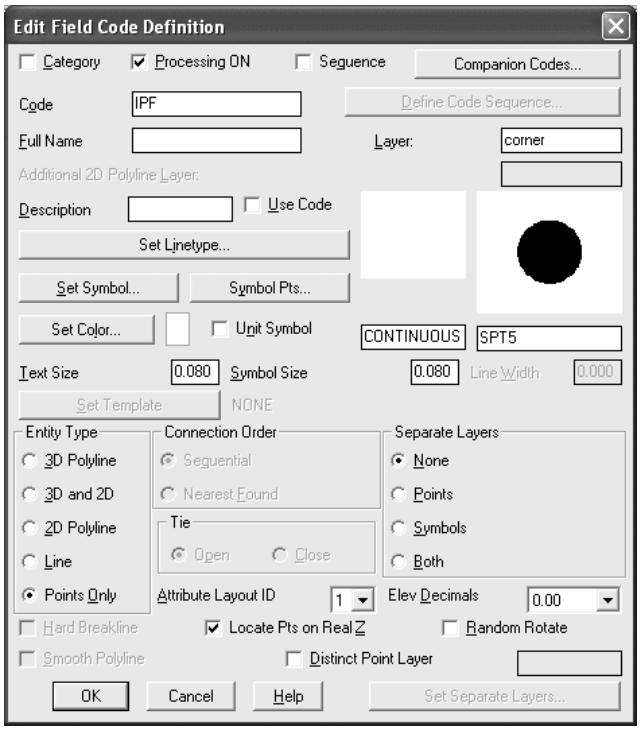
- **Line Width**: This controls the width for the linework. Only applies to 2D polylines.

- **Distinct Point Layer**: When this toggle is selected, the line work is created in the layer defined in the Layer field and the points are created in the specified distinct point layer. For example, you could have DRIVEWAY for linework and DRIVEWAY_PNT for the points.

- **Entity Type (3D and 2D)**: This option allows polylines to be drawn as both 3D and 2D. When clicked on, the "Additional 2D Polyline Layer option, near the top of the standard Edit dialog, allows you to place the 2D polyline on a different layer than the 3D polyline. A curved polyline coded with the PC or equivalent "start curve" code, for example, would plot with a true arc for the 2D polyline and with a series of 3D interpolated vertices through the arc in the case of the 3D polyline.

2 **Add** - The new code definition is inserted in the list in the position after the currently selected one. If none are selected for positioning, the new code is placed at the top. Only one code definition may be highlighted before running this routine.

3 **Copy**- The "Copy" command requires that you first select and highlight a single code, and then it captures all the settings in the standard Edit dialog, but leaves the code name blank requiring entry of a new code name. It might be used to add a new IPF (iron pin found) by borrowing from IP, changing nothing but the symbol, as shown below:



4 **Cut** - This command will remove the highlighted code definitions from the list and puts them in a buffer for retrieval with Paste.

5 **Paste** - This command will insert the code definitions put in the buffer by the Cut command. These codes will be inserted after the row of the currently highlighted code or at the top.

6 **Search** - Allows you to search for a specific code in the list.

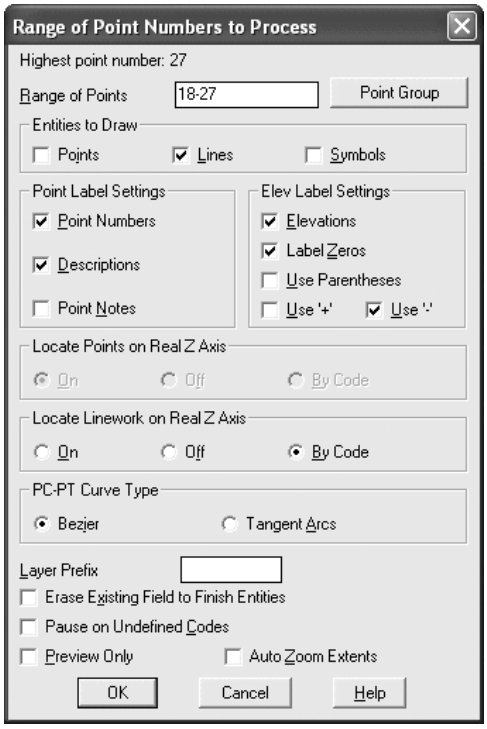
7 **Save** - Saves the Field to Finish code list (.FLD) file.

Set/Process Data File

1 **Set CRD File** - This command allows you to specify a coordinate (.CRD) file to process.

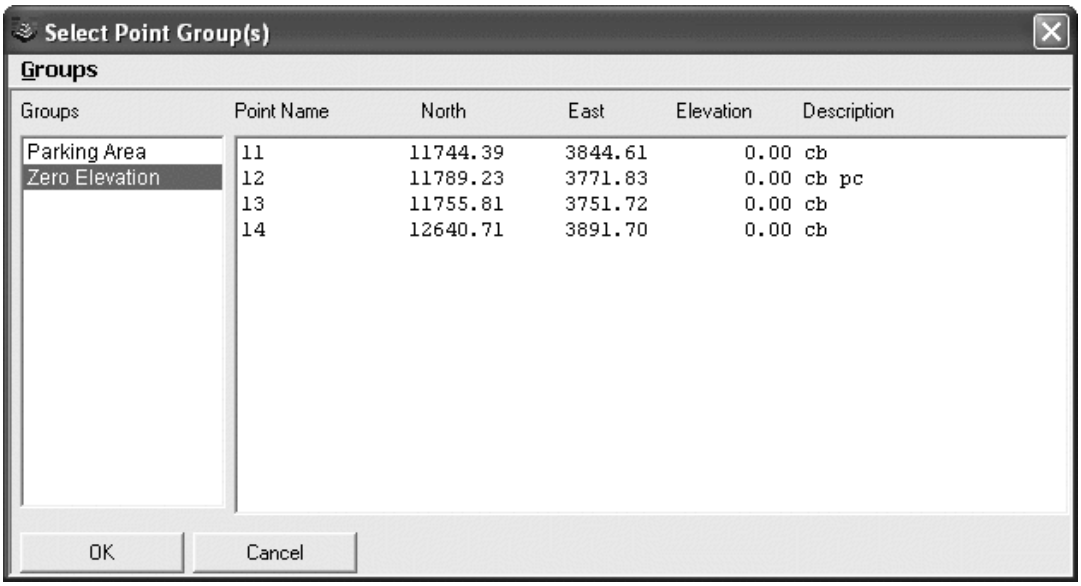
2 **Edit Points** - This command opens the Edit Points spreadsheet editor. See Edit Points for more details.

3 **Draw** - The Draw Option leads to a dialog that controls the range of points to process. With Carlson TakeOff, this dialog also controls whether points only, lines only or lines and points are plotted. If you choose to plot lines only, that selection will be the default until changed. The Draw dialog, within Field-to-Finish, is shown below:

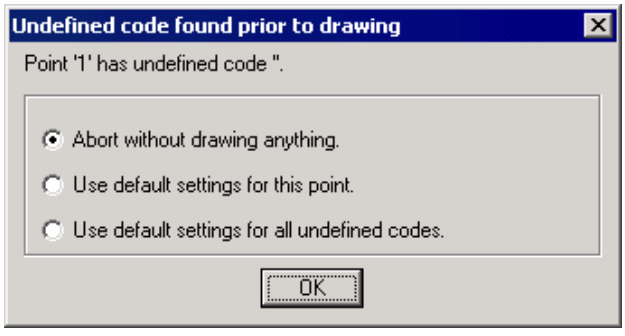


- **Range of Points**: Specify the range of points to draw.

Point Groups: Point Groups are another way of defining a range of points to plot. Point Groups can be defined using the Point Group Manager under the Points pulldown menu, and include points sharing certain descriptions, elevation ranges, locations on the screen, etc. Here is an example of a Point Group selection set (points with zero elevation):



- **Point Label Settings**: Specify whether you want Field to Finish to label the Point Numbers, Descriptions, and/or Points Notes which are contained in the note (.NOT) file that is associated with the coordinate (.CRD) file.
- **Elev Label Settings**: Specify the elevation labeling options. The Label Zeros option will label the elevations of points with z=0 Use Parentheses will place parenthesis around the elevation text.
- **Locate Points on Real Z Axis**: Choose between locating all the points at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes.
- **Locate Linework on Real Z Axis**: Choose between locating all the linework at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes.
- **PC-PT Curve Type**: Sets the method for drawing curves with more than 3 points. The Bezier option draws a smooth polyline through all the curve points. The Tangent Arcs method draws multiple arcs with arc end points at each of the curve points. These arcs are tangent to the preceding line segment.
- **Layer Prefix**: Optional layer prefix added to all entities drawn with Field to Finish.
- **Erase Existing Field to Finish Entities**: When checked, this option will erase from the drawing any old entities created by previous Field-To-Finish runs before drawing the new entities.
- **Pause on Undefined Codes**: When checked, Field to Finish will pause if it encounters a description that is not defined in the code table.


Undefined code found prior to drawing
Point '1' has undefined code ''.
Abort without drawing anything.
Use default settings for this point.
Use default settings for all undefined codes.
OK

- **Abort without drawing anything**: This lets you stop to correct the code table.
- **Use the default settings for this point**: This default is to draw a point in the "MISC" layer with no linework. To

set your own default, define a code called "SC_DFLT".

- **Use the default settings for all undefined codes**: Continue processing and use default code for all undefined codes.

A good way to check the data file for unmatched descriptions is to use the Print Table command and choose the Data Points and Distinct Code options. This command will print the different codes in the data file and identify any undefined codes.

- **Preview Only**: When checked, this option will temporarily draw the points and linework and allow you to review it with zoom and pan.
- **Auto Zoom Extents**: When checked, this will force a zoom extents after Field to Finish is done.
- **Draw Points Only**: This command creates only the points.
- **Draw Lines Only**: This command creates only the line work.

Prerequisite: A data file of points with descriptions.

Keyboard Command: fld2fin

Spreadsheet Edit Points

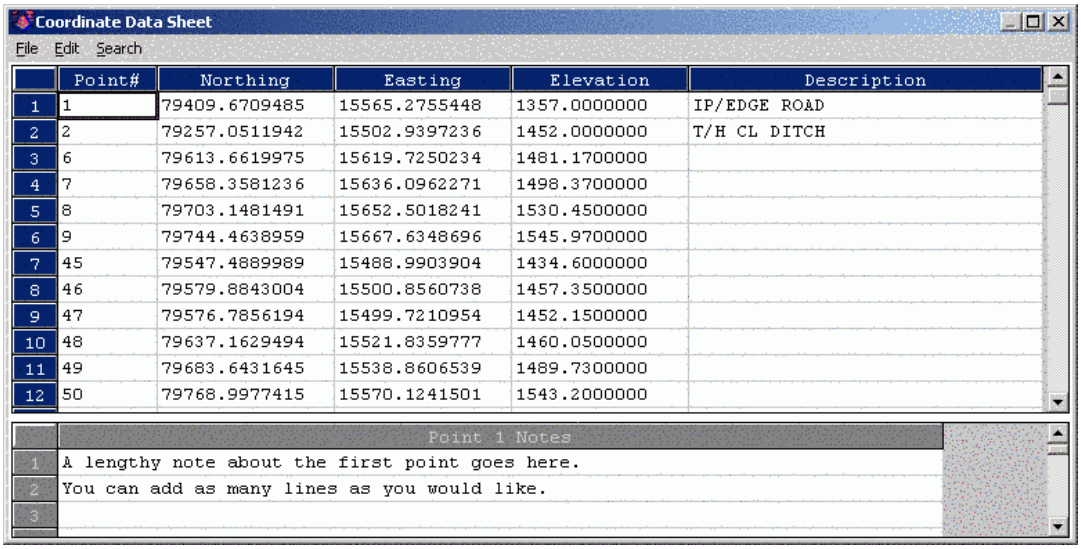
Function

This command edits point data in the current coordinate file. The current coordinate file can be set with the Set Coordinate File command. Edit Points shows all the points in the coordinate file. New points can be added and points can be deleted by using the Insert and Delete keys.

This tool also lets you edit notes associated with each point. While the standard point description is limited to 32 characters, the drawing notes are not. When you click on a given point, you can add numerous lines of notes about that point in the bottom of the dialog. Keep in mind that these notes are stored in a separate file with the extension ".not" having the same name as the CRD and residing in the same folder.

Keyboard Command: EDITPT

Prerequisite: None



On-Screen Edit Points

Function

This command will edit the attributes of a TakeOff point such as the symbol type, point number, elevation and description. When this command is invoked, the command line will prompt the user: **Select point to edit (Enter to end)**. At this point, you can select any part of the point including the symbol, elevation, point number or the description. Next, a dialog will appear as shown.

To change the symbol, either type in a new symbol name in the edit box, or choose the "Select Symbol" button where you can choose from a list of symbols. To change any of the other properties of the point, simply change or replace the contents of the edit box with the new information. If you change the point number to a number that already exists in the current CRD file, and point protect is ON, you will be prompted **[O]verwrite w/new coordinates, overwrite [A]ll, or use number <1000>:.** You can choose to use the next available point number in the CRD file (this is the default) or overwrite the point number. The properties that you modify with this command will update the current CRD file and the screen entities.

You may also choose to use the AutoCAD *DDATTE* command to change the attributes of a point. If you do this, then the CRD file will not be updated and if you change the elevation attribute, the point will not change its current Z location.

Prerequisite: TakeOff points

Keyboard Command: EDITPNT

Scale_Point_Attributes

Function

This command will scale point attribute text (number, elevation and descriptions) and point symbols up or down in size. The routine prompts for a scale multiplier and a selection set of objects. If you want to enlarge, enter a value

greater than one. If you want to reduce, enter a decimal fraction such as .5. This would reduce the text size by 50%. This command is very useful if you have set up your drawing for one plotting scale and decide to change to a new plotting scale. This command has the added benefit that it will adjust the point attributes and symbols to a new screen twist angle.

Prompts

<>**Scaling Multiplier <0.5>:** *2.5* This response would enlarge the point attributes and symbols by 250 percent.
This command will adjust the point attributes to the current screen twist !
Select Takeoff Point Attributes & Symbols to Enlarge/Reduce.
Select objects: *C*
First corner: *pick a point*
Other corner: *pick a point*
Select objects: *press Enter*
Number of entities changed 30

Keyboard Command: pntenl
Prerequisite: points</>

Resize_Point_Attributes

Function

This command sets the size of the selected point attributes (point number, elevation, description) and point symbols. This command is similar to Scale Point Attributes but instead of scaling the size by a factor, all the select points are set to the same specified size.

Prompts

<>**Enter point attribute and symbol size <4.0>:** *press Enter*
Select SurvCADD Point Attributes & Symbols to resize.
Select objects: *pick the point entities*
Finding Takeoff Point Attributes ....
Number of entities changed> 39

Keyboard Command: sizepnt
Prerequisite: points
</>

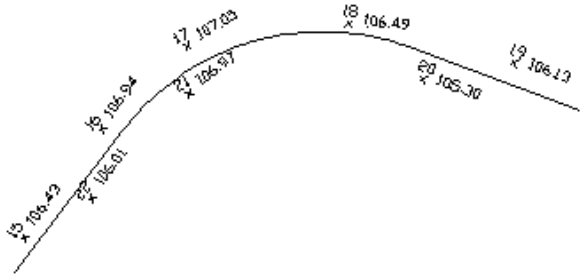
Twist Point Attributes

Function

This command will rotate the orientation of the text of Carlson point attributes (point #, elevation, description) and point symbols. The Twist Screen option aligns the point attributes to appear horizontal in the current twist screen. The Azimuth option allows you to enter an azimuth or pick two points to align the point attributes. The Entity Segment option aligns the point attributes by the selected line or polyline segment in the direction the entity is drawn. The Follow Polyline option aligns the point attributes by the polyline segment that is closest to the point.

Prompts

Twist by [<Twist screen>/Azimuth/Entity segment/Follow polyline]? *F*
Select reference polylines to follow. *pick a polyline*
Select objects: 1 found
Select objects:
Select points from screen, group or by point number [<Screen>/Group/Number]? *select Enter*
Select Carlson Software points.
Select objects: *pick the Carlson point inserts*



Point attributes aligned by Follow Polyline option of Twist Point

Keyboard Command: twistpts
Prerequisite: None

Erase Points

Function

This command erases Carlson points from the drawing. The points to erase can either be selected from the screen or specified by point number. Erasing a Carlson point will erase the point symbol, point attributes, and point node. The points may optionally be erased from the coordinate file. As long as the points are not deleted from the coordinate file, they can be redrawn with the Draw-Locate Points command.

Prompts

1 Select points from screen or by point number (Screen/<Number>)? **Enter**

2 Point numbers to erase: **5**

3 Delete points from coordinate file (Yes/<No>)? **Y**

Erased 1 point from CRD file.

Erased 1 point from drawing.

Prerequisite: Carlson points.

Keyboard Command: DELPT

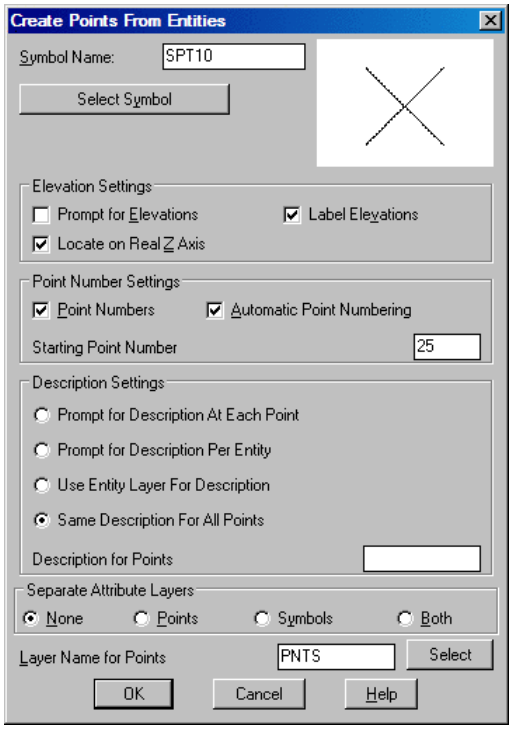
Create Points from Entities

Function

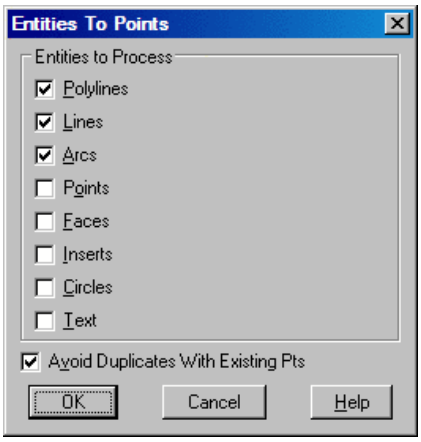
This command will create Carlson points on selected entities. The points are stored in the current coordinate (.CRD) file and drawn on the screen. For arcs and polylines with arc segments, points are created at the radius points of the arcs as well as the PC and PT.

In the first options dialog, there are settings for the point attributes. To have points obtain their elevation from the selected entities, unselect the **Prompt for Elevations** toggle and select the **Locate on Real Z Axis** toggle. After you have specified the point options, a secondary dialog appears which allows you to specify the entity types to process. Under the **Description Settings, Prompt for Description At Each Point** will prompt you at the command line for a description for each individual point. **Prompt Per Entity** will ask you for a description per each highlighted entity. **Use Entity Layer for Description** will assign the layer name to the description. When Entity Layer for Description is checked, the layer name of the entity will be used as the description for the created point. **Same Description For All Points** will prompt you for a single description for all points.

The second options dialog has processing settings. When **Avoid Duplicates with Existing Pts** is checked, this routine will not create a point if a point with the same coordinates already exists in the current coordinate (.CRD) file.



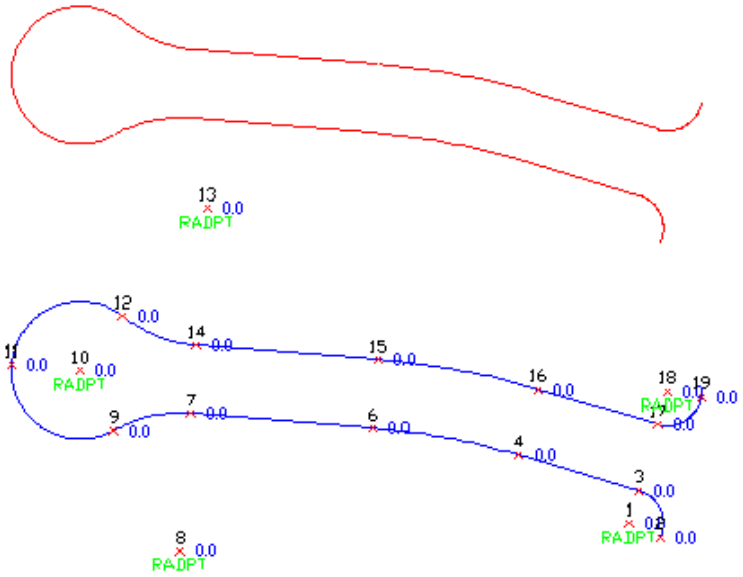
Routine begins with this dialog



After clicking *OK* on the first dialog

Prompts

Create Points From Entities Dialogs Choose settings
Select arcs, circles, faces, points, text, lines and polylines.
Select objects: *pick entities*



Before and after using Create Points from Entities. Points are created at each endpoint and radius point.

Keyboard Command: autopnts
Prerequisite: drawing entities

Coordinate File Utilities

Function

This command allows you to manipulate the coordinates stored in a coordinate (.CRD) file. One of the most important commands is the Update CRD File from Drawing which allows you to update the file after editing the drawing with commands such as *Erase*, *Move*, *Rotate* or *Change Elevations*. Another handy option is the *Draw Entities by Point Number* which allows the user to input point number ranges and plot Lines, Arcs, Polylines or 3D polylines. Coordinate files have either numeric or alphanumeric point numbers. Alphanumeric point numbers consist of nine or less digits and letters (i.e. point number 7A). The type of point number format is displayed at the top title bar of the dialog shown on the next page.

Coordinate File Utilities Options

Open CRD File: Allows the user to switch to another file. When you exit Coordinate File Utilities this will be the current file that you work with in TakeOff.

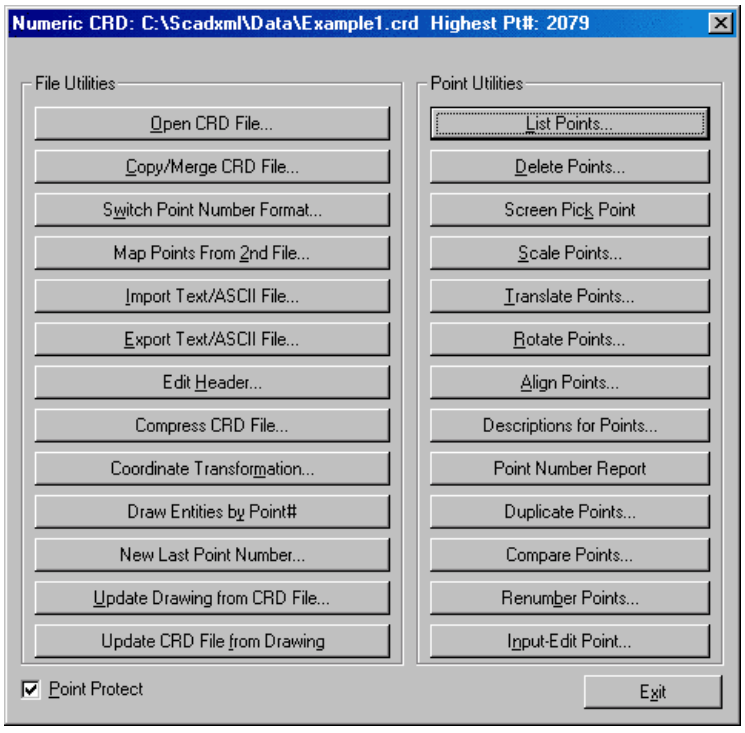
Copy CRD File: Copies a coordinate file to another file name. This can be used to make a backup of your coordinate file.

Switch Point Number Format: Allows you to convert the current CRD file from numeric format to alphanumeric format.

Import Text/ASCII File: This routine converts point data from a text file into the current coordinate (.CRD) file. See the *Import Text/ASCII File* command in this chapter for more information.

Export Text/ASCII Text File: This routine outputs point data from the current coordinate (.CRD) file to a ASCII Text file. See the *Export Text/ASCII File* command in this chapter for more information.

Edit Header: Enter or edit the job information associ



ated with the coordinate file. Fields include Job Description, Job Number and Job Date.

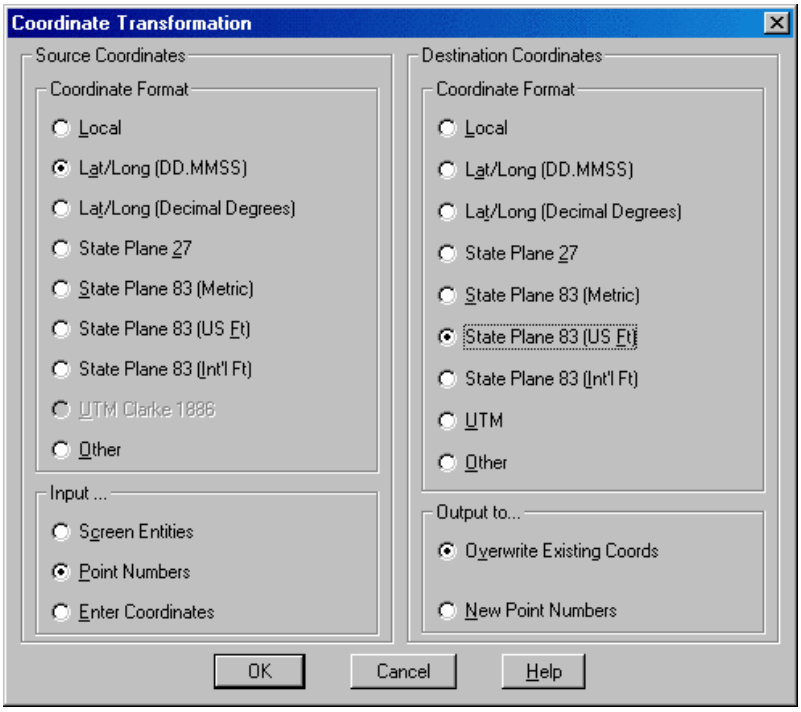
Compress CRD File: Removes unused point numbers by renumbering high point numbers into the unused spaces. For example, for an original file with points 1,2,105,107,108,109 would be compressed to 1,2,3,4,5,6.

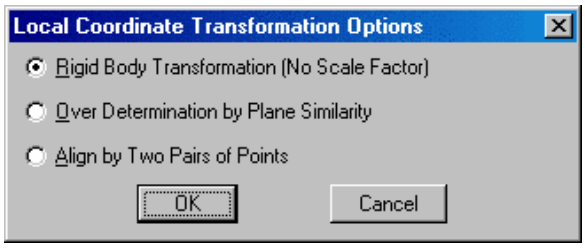
Coordinate Transformation: Transforms coordinates between local, state plane 27, state plane 83, latitude/longitude, and Universal Transverse Mercator (UTM). Works on individually entered coordinates, by range of point numbers and with on-screen entities. For converting between state plane 27 and 83, TakeOff calls upon NADCON from the National Geodetic Survey to apply the latitude/longitude adjustment. The NADCON program, ndcon210.exe, is stored in the TakeOff EXEC directory.

The Enter Coordinates input option transforms one coordinate at a time. The coordinates can be typed in or use the Input Point Number option. Output Point Number is an option to store the results in the coordinate file.

When transforming a local coordinate system, there are three methods as shown in this dialog. The Align by Two Pairs of Points option uses two pairs of source and destination coordinates. The first pair defines the translation as the difference between the source and destination northing and easting. This destination point is also the pivot point for rotation. Rotation can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply scaling. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.

The other local transformation options are used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information. Over Determination by Plane Similarity is used to find the least squares best fit





transformation for all the given source and destination points. Besides doing a translation and rotation, this option will also scales the points during the transformation. The Rigid Body Transformation also does a best fit least squares transformation but applies only translation and rotation with no scale.

When running TakeOff with AutoCAD Map, the Coordinate Transformation dialog has an option called "Other" which activates all the AutoCAD Map transformations.

Draw Entities by Point#'s: Draw Lines, Arcs, 3DLines, Polylines or 3DPolys by defining a range of point numbers.

Example Prompts:

Enter Menu Option? <L>: *P*

Plot Entities by Point Number

Type of entity, Arc/Polyline/3dpoly/2dline/Exit/<Line>: *P*

This response causes the program to plot polylines.

Example: '1*4-7-10*12-5-8' would draw lines from point number's 1 through 4 then to 7, to 10 through 12, then to 5 to 8. (limit 132 characters)

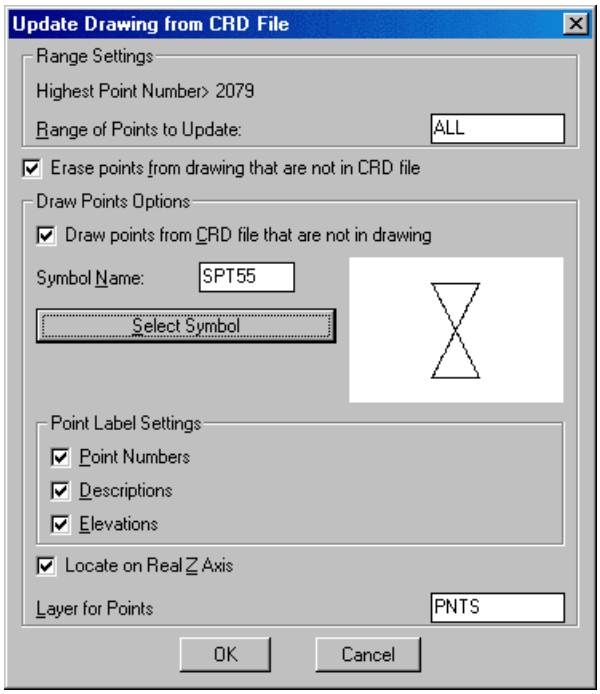
Undo/<Enter point numbers or ranges>: *1*10-20*30*

The program draws a polyline from point number 1 through 10 to point number 20 through 30.

New Last Point Number: This option sets the highest point number in the CRD file. All points above this number are erased.

Update Drawing from CRD File: This function updates the position of TakeOff points in the drawing to match the position stored in the coordinate file. This command also has options to erase and draw points. For the erase option, point are erased from the drawing if the point number does not exist in the coordinate file. For the draw option, if a point number in the CRD file does not exist in the drawing, then this point is drawn using the settings from the dialog. The number of points modified, erased and drawn is reported at the end of the command.

Update CRD File from Drawing: This function allows you to select all or some of the points in the drawing and add or update them to the .CRD file. The points can be filtered with AutoCAD's Select Objects: selection mechanism and/or wild card matching of the point descriptions. The Update Point Descriptions option determines whether the point descriptions from the drawing will be stored to the CRD file. Use this command to update the file after a global edit such as *Move*, *Rotate*, *Renumber Points*, *Change Elevations*, *Erase*, etc. This routine directly reads Leica (Wildsoft), Softdesk, Geodimeter, InRoads, Land Development Desktop, and Eagle Point point



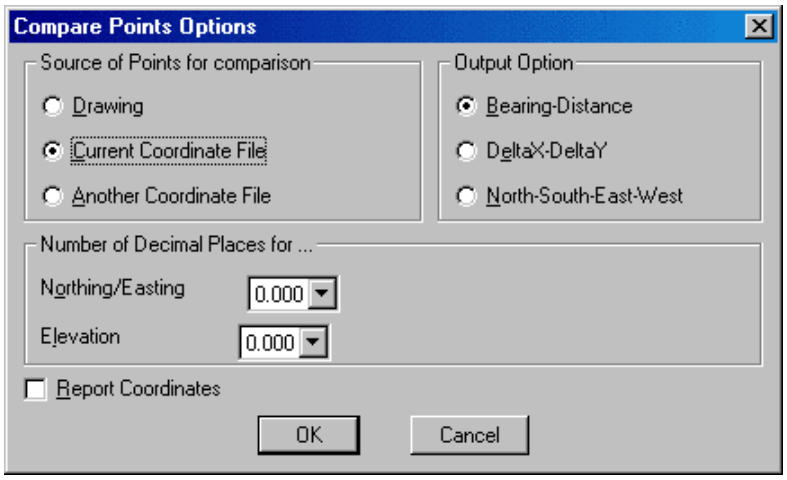
blocks.

Compare Points: This function compares the coordinates in the .CRD file with either the coordinates for the matching point numbers in the drawing file, with matching point numbers from another CRD file or with different point numbers from the same CRD file. A report is created for any differences that shows the point numbers and the differences. The difference can be reported as a bearing and distance between the two points, as distance North/South and East/West or as the delta-X and delta-Y. There is an option whether to include the point coordinates in the report.

List Points: List the points stored in the .CRD file. See the *List Points* command in this chapter for more information.

Delete Points: Deletes points in the file by defining a range of point numbers to delete or by picking a polyline that defines a perimeter from which points inside or outside the perimeter are deleted.

Screen Pick Point: Pick a point on the graphics screen and add or update it's coordinate values



Compare Drawing to CRD File> northpark.crd

Point# North South East West Elevation

_____

1125 0.28 0.30 0.00

1124 1.97 1.73 0.00

1123 1.99 1.45 0.00

1085 1.53 1.45 0.00

1084 1.53 1.73 0.00

1083 0.17 0.31 0.00

Example North-South-East-West format Compare Points Report

Point# Northing Easting Elevation HzDiff Bearing

1125 5000.14 4939.65 89.97 Drawing

1125 4999.86 4939.35 89.97 CRD File

0.41 S 47d36'46'' W

1123 5079.43 4941.93 90.94 Drawing

1123 5077.44 4940.48 90.94 CRD File

2.46 S 36d02'36'' W

1083 4999.03 4875.04 89.95 Drawing

1083 4999.21 4874.73 89.95 CRD File

0.35 N 60d26'44'' W

Example Bearing-Distance with Coordinates format Compare Points Report

to the .CRD file. This command does not plot a point, point attributes or point symbol. Use the command *Draw-Locate Points* command to do this.

Scale Points: This option multiplies the point northing, easting, and elevation by the scale conversion factor. You can use this routine for metric-english conversion. See the *Scale Points* command in this chapter for more information.

Translate Points: This option translates a range of points based on entered delta x and delta y, entered coordinates or translation point numbers. See the *Translate Points* command in this chapter for more information.

Rotate Points: This option rotates a range of points based on entered degrees or rotation, entered azimuths, entered bearings or rotation point numbers. See the *Rotate Points* command in this chapter for more information.

Align Points: This option does a translate based on a source point and destination point and then rotates to align the first source point and a second source point with the first destination point and a second destination point. See the *Align Points* command in this chapter for more information.

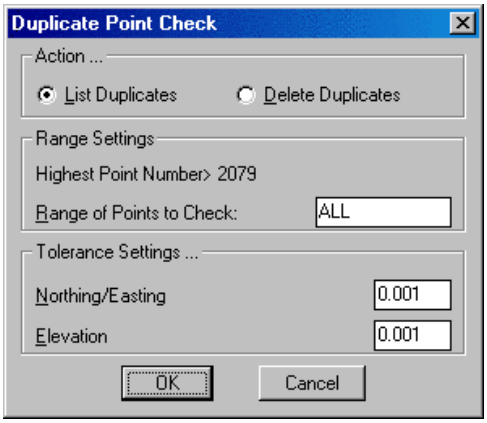
Duplicate Points: This function searches the CRD file for points with the same northing, easting and elevation. The tolerances for considering points to have the same coordinate are set in the dialog separately for northing/easting and elevation. To be counted the same coordinate, both the northing/easting and elevation must be within the tolerance distance. The duplicate points can be erased or only reported. For the erase option, the first point number is kept and any higher point numbers with duplicate coordinates are erased from the CRD file.

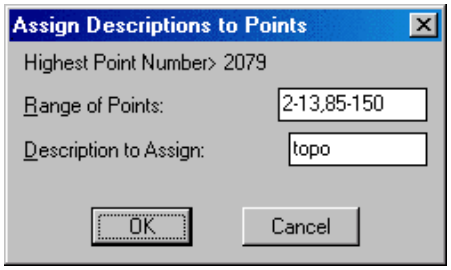
Point Number Report: This routine list the used and unused point numbers in the .CRD file.

Renumber Points: This option renumbers points in the user-specified range starting from a new point number. The old point numbers are erased. The condense points will renumber such that there are no unused point numbers in the renumbered range. Otherwise the spaces between the points is maintained. In the example shown, renumbering 1-25 with points 1,2,24,25 to starting point number 101 will result in points 101,102,103,104 if condense is on or 101,102,124,125 if condense is off.

Description for Points: This option sets the point description field with the user-specified text for a range of point numbers.

Input-Edit Point: Enter or edit the coordinate values or the





description of a point. The Notes section is for adding optional point notes which are additional point descriptions. The standard description field is limited to 32 characters. Under notes, any number of lines of text can be assigned to the point. A list box shows the lines of notes. To add a note line, pick a blank line in the list box and then type in the note in the edit box belong the list box and press Enter. To edit a note, highlight the line in the list box and edit the text in the edit box.

Map Points from 2nd File: This routine adds point to the current CRD file from points stored in a second CRD file. The points to copy are specified by numbers one at a time.

Destination Point Number: *55* This is the point number to create in the current CRD file.

Source Point Number: *25* This is the point number to copy from the second CRD file.

Point# Northing Easting Elevation

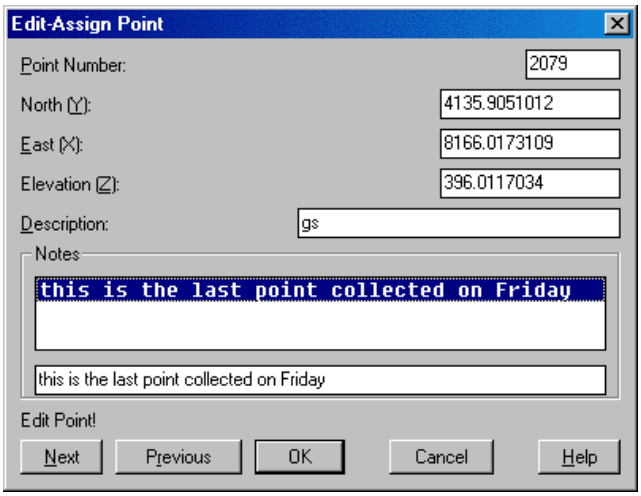
25 52.516 13.328 0.000

Destination Point Number: *Press Enter to end*

Point Protect Toggle: Toggles point protection on and off.

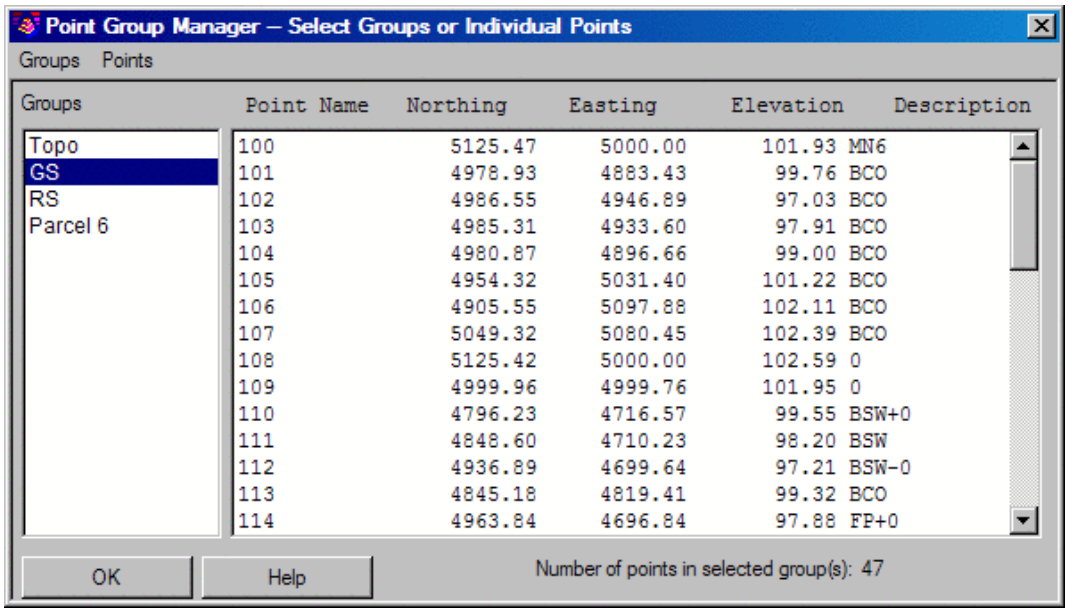
Keyboard Command: CFU

Prerequisite: None



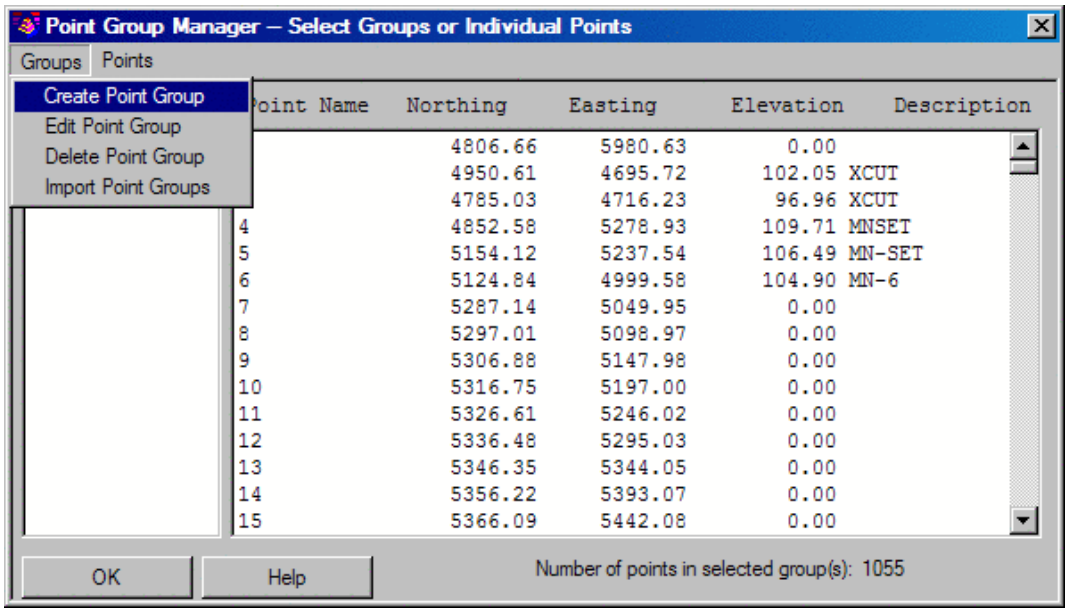
Point Group Manager

This command is used to create point groups based on inclusion and exclusion filters. The manager can perform various functions on these point groups. Also point groups can be referenced by group name in other commands such as Field to Finish and Data Collection.



Groups Pulldown

Create Point Group: This option creates point groups. When selected, the New Point Group dialog box is displayed.



Group Name is the name of Point Group to create.

Description is the description of Point Group to create.

Use the **Include Tab** to define the filters to be applied when creating the point group. Inclusion rules are applied before the exclusion rules.

When **Include All** is toggled on, all points in the coordinate file will be included in the selection.

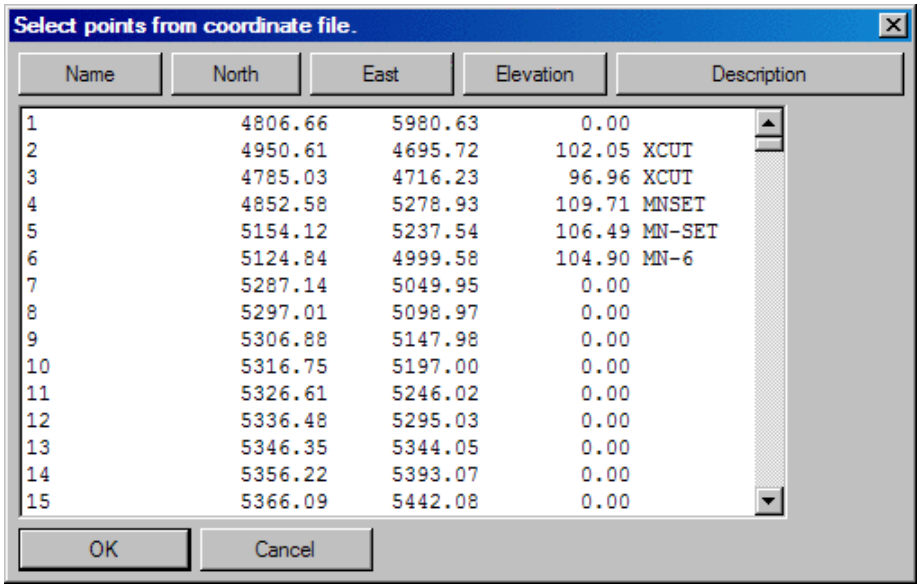
When **Point List** is toggled on, an option of defining the point list must be selected.

DWG: Select allows for manual selection of the points to include from the drawing. The points must be drawn on the screen prior to using this option. All standard AutoCAD selection tools, are available for selection of the points.

DWG: Add Within Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. Points must be drawn to the screen prior to using this option.

DWG: Add Within Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. Points must be drawn to the screen prior to using this option.

CRD: Select allows for manual selection of the points to include from a point list. Standard window selection tools are available for selecting the points to include.



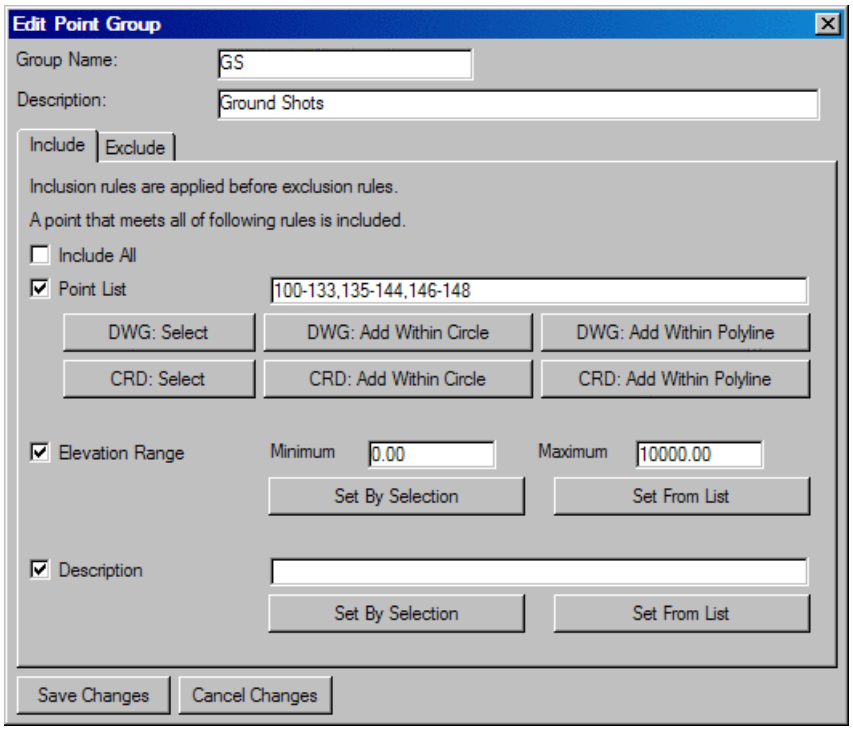
CRD: Add Within Circle allows for selection of the points to include by a user defined circle. The circle is defined by specifying the center and radius of the circle. The radius can be defined by entering in a numeric value or by picking on the screen. The points do NOT have to be drawn to the screen prior to selection.

CRD: Add Within Polyline allows for the selection of points to include by referencing a closed polyline. All points located within the closed polyline will be included in the selection. Prompts for the inclusion polyline and the exclusion polyline will display. The inclusion polyline limits of the selection area. The exclusion polyline defines the area to exclude within the inclusion polyline. The points do NOT have to be drawn to the screen prior to selection.

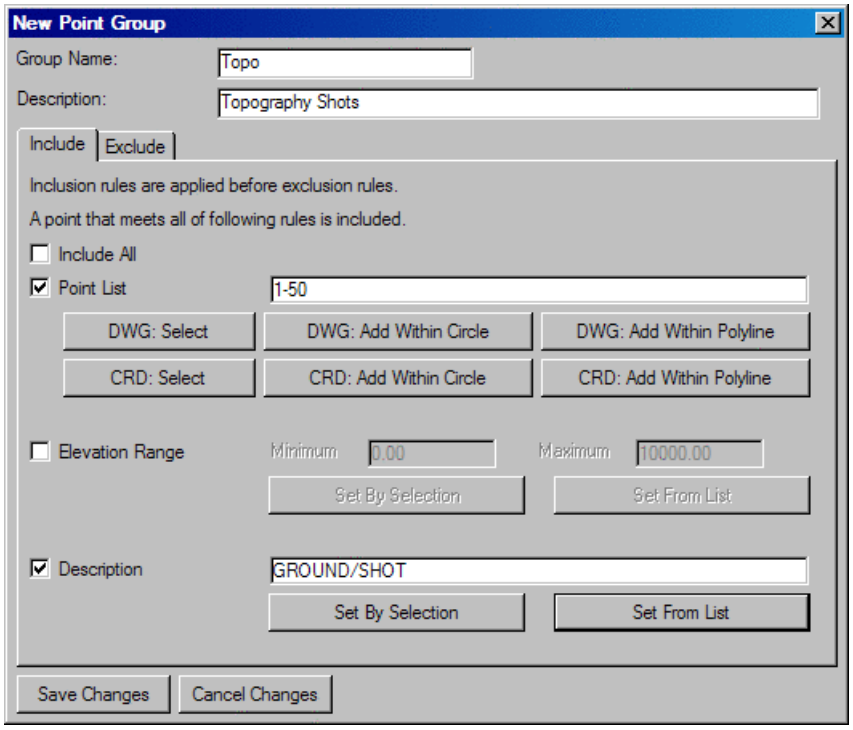
Elevation Range allows for the selection of points within a specified elevation range to be included in the group. The minimum and maximum elevations can be entered manually in their respective data fields. The minimum and maximum values can also be specified by the Set By Selection and Set From List options.

Set By Selection allows for selection of points to include in the group from the drawing. The points must be drawn to the screen prior to using this selection method. Standard AutoCAD selection methods are available.

Set From List allows for selection of points to include in the group from a point list. Standard Windows selection tools are available with this option.



The **Description** option allows for a selection of points to include based upon the description of the point. The description to filter for can be entered in the data field or by using the Set By Selection and/or the Set From List options described above.



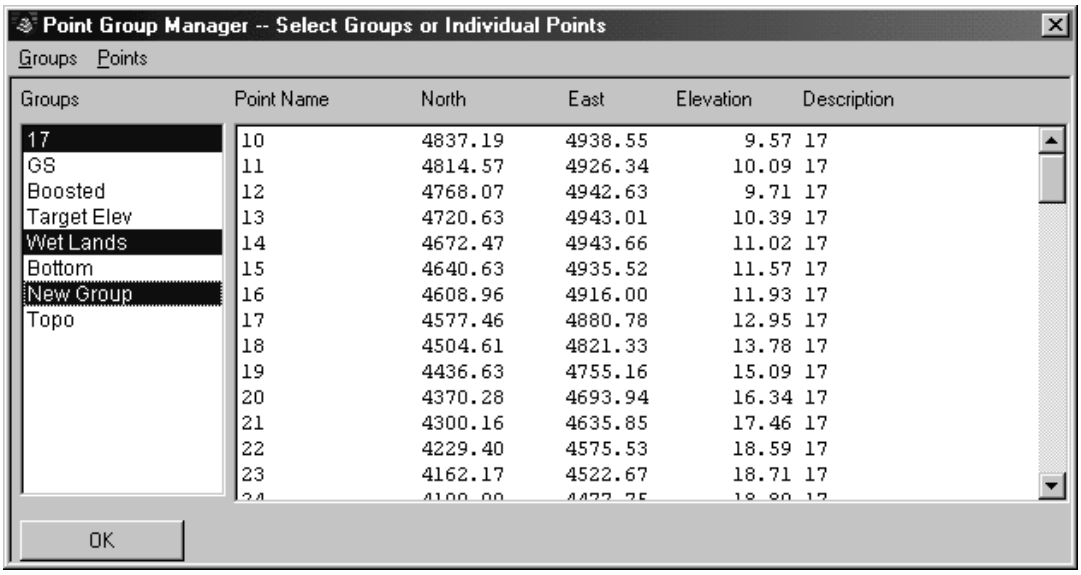
The **Exclude Tab** allows for defining rules that pertain to the points to be excluded from the Inclusion selection. After defining the inclusion rules for the group, the options on the Exclude tab can be used to filter for points to exclude from the group. For example, if the inclusion rules call for all points within the elevation range of 8 to 12,

an exclusion rule can be set to exclude the points on elevation 9 or with the description tree. The options on this tab work exactly like the options on the Include tab. Please refer to the Include tab definitions for further instruction.

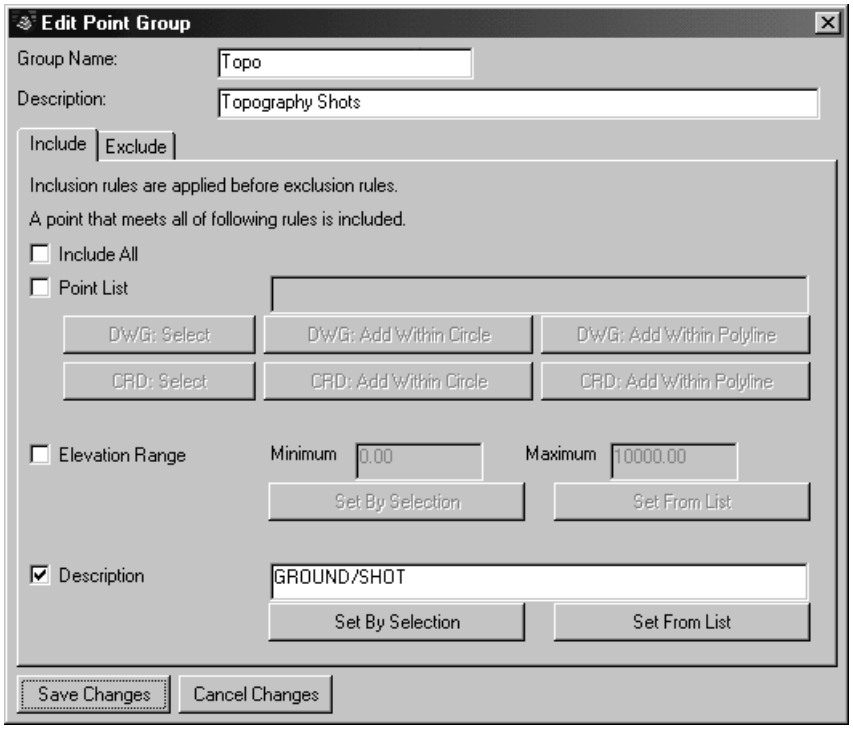
Save Changes saves the point group to the group name specified based upon the Inclusion and Exclusion rules specified.

Cancel Changes discards specified rules and changes and goes back to the Point Group Manager dialog.

Edit Point Group allows for editing of existing point groups. From the list of available groups, highlight the group or groups to edit. When complete with the first group, if more than one is selected, selecting the Save Changes option will save the changes to the active group and switch to the next group in the selection set.



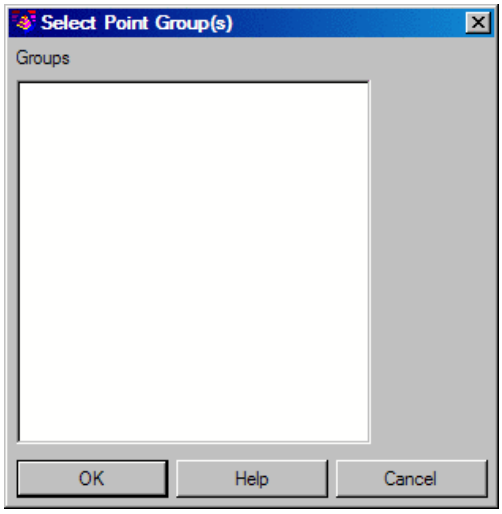
From the Groups pulldown, select Edit Groups, the Edit Group dialog box will now appear.



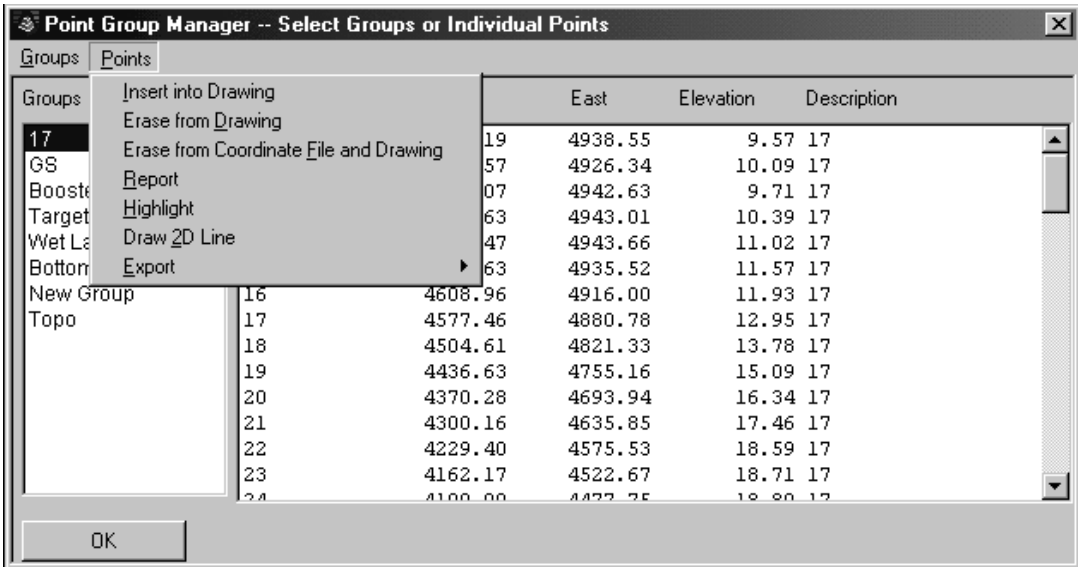
See Create Point Groups for further definitions of the available options.

Delete Point Groups deletes specified groups for the existing group list. One or more groups can be deleted at one time.

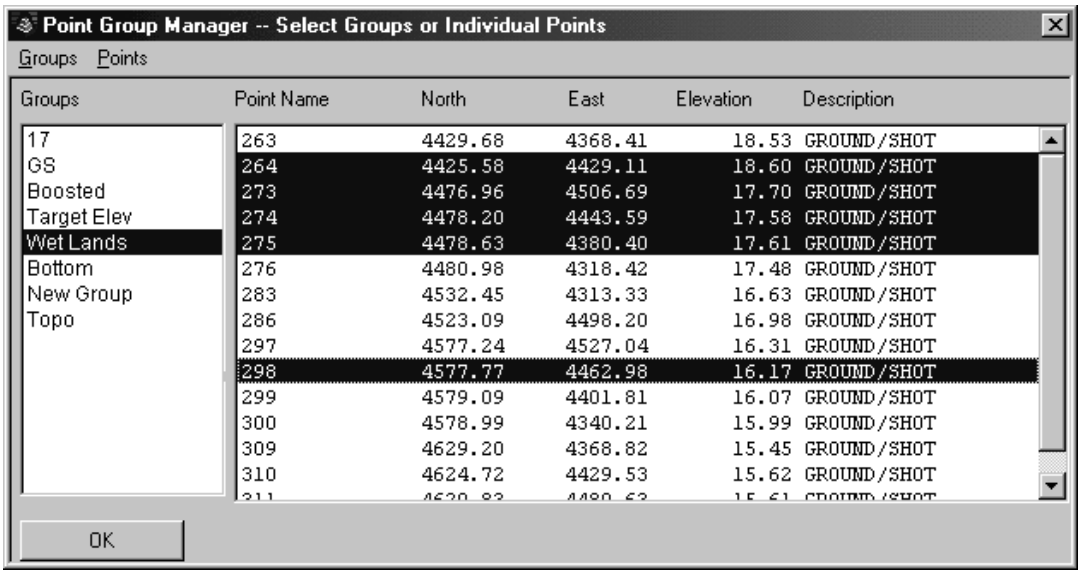
Import Point Groups allows for importing filters from point group manager settings of other coordinate files. This is a useful option when coordinate files are going to contain same point group names with the same filters. This option only brings in the filters into the point group manager, it does not import actual points into the coordinate file by group name. Existing points in the active coordinate file that meet the filter definitions of the imported point groups will automatically be added to the corresponding group.



Points Pulldown



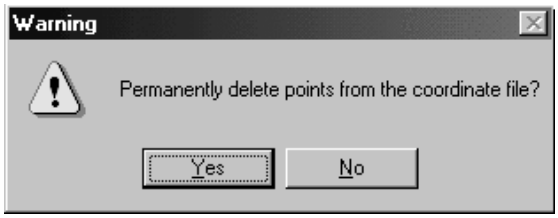
The **Insert into Drawing** option draws the points in the group in the drawing. Individual points or point ranges can be selected from the group to be erased from the drawing. For example points 264-275 and point 298 contained in group Wet Lands are tagged to be erased from the drawing in the following figure.



The symbol to be used and the attribute layout are determined by the Point Default Settings. The symbol size and the point attribute size are determined by the settings in the Drawing Setup routine.

Erase from Drawing erases specified point group/groups or specified points from within the group from the drawing.

Erase from Coordinate File and Drawing erases the points in the specified group/groups or specified points from within the group from the drawing and will also permanently delete the points from the CRD file. You will be prompted with a warning as follows:



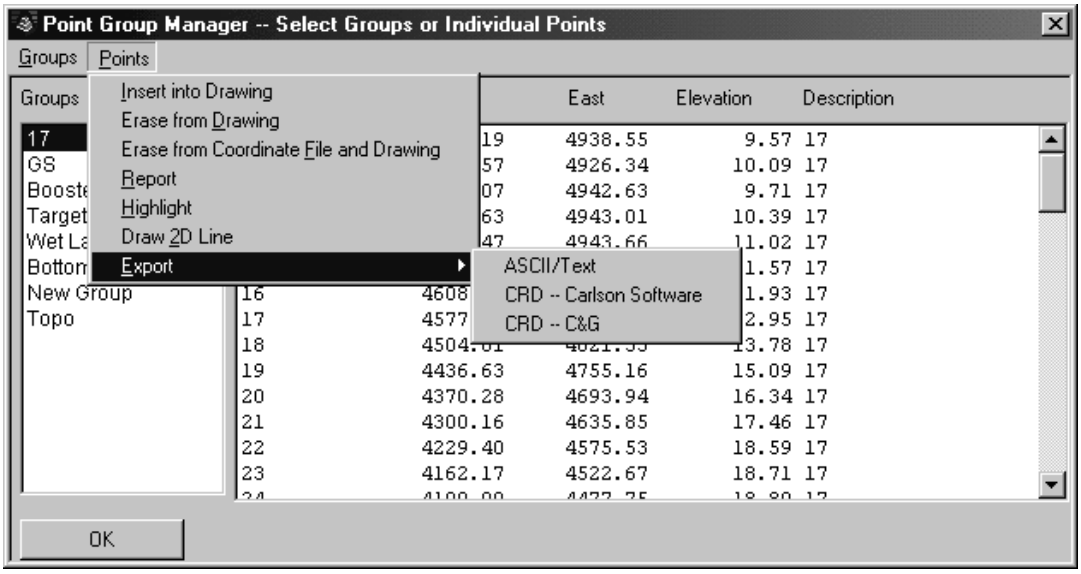
Selecting **Yes** will complete the command and erase the points from the screen and also the coordinate file. Selecting **No** will cancel the command leaving the drawing and the coordinate file unchanged.

The **Report** option will generate a point list of the points contained in the selected group/groups or specified points from within the group.

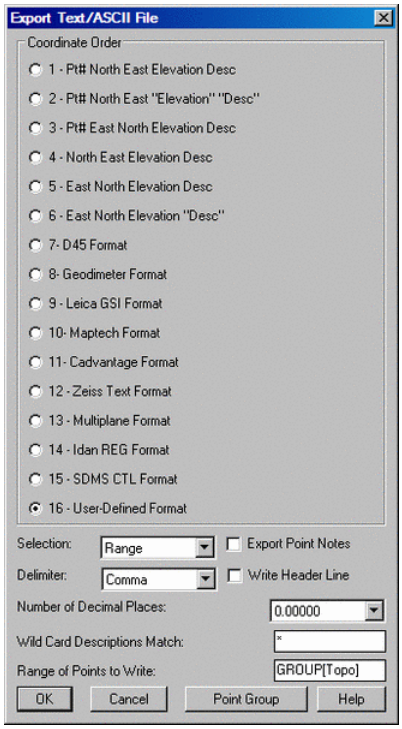
The **Highlight** option highlights the specified objects in the drawing. This makes them distinguishable from the other points on the screen.

The **Draw 2D Line** option draws a 2d polyline between the points contained in the group/groups or between specified points in a group.

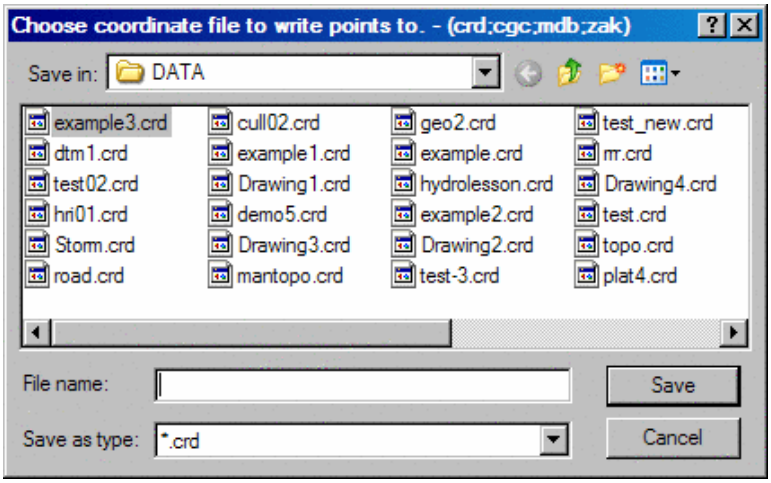
The **Export** command exports the selected group/groups or the specified point(s) or range of points from within the group to various formats. The available formats are ASCII/Text, Carlson Software CRD and C&G CRD files.



When **ASCII/Text** is selected, the Export Text/ASCII File dialog box will be displayed. Please refer to the Export Text/ASCII File section of the manual for more information.



The **CRD-Carlson software** command writes the selected group/groups or the specified point(s) or range of points within the group to a new Carlson formatted CRD file.



Specify the file name of the CRD file to create and press save.

CRD-C&G writes the selected group/groups or the specified point(s) or range of points within the group to a new C&G formatted CRD file.

Specify the file name of the CRD file to create and press save.

Keyboard Commands: pgm
Prerequisite: A coordinate file

Interpolate Points

These command interpolates points by either: Point on Arc, Divide Between Points, Divide Along Entity, or Interval Along Entity.

Function

This command locates a point on an arc. You can select an arc entity, an arc polyline segment or enter three points to define an arc. After the arc is defined, the screen preview arrow shows the occupied point and the distance to solve for is entered. The command then displays the curve information and locates/inserts a point symbol at the computed point. When prompted for the distance, use a positive value if the distance is from the 1st endpoint (PC the one highlighted by the screen preview arrow) and a negative value if from the 2nd endpoint (PT).

Prompts

Define arc by, Points/<select arc or polyline>: *pick arc or polyline arc segment* Pick a point on the arc somewhere near it's midpoint. The preview arrow points to the 1st endpoint.
Precede distance with minus sign if distance from 2nd endpoint.
Distance along arc from 1st point: *100*
The command then plots a point at the computed distance.

Keyboard Command: ptarc
Prerequisite: None

Function

This command divides the distance between two points and inserts one of the point symbols at the specified distances. It can also interpolate elevations (to interpolate the elevations, the points picked must be at their real Z axis elevation).

Prompts

Interpolate elevations [Yes/<No>]? *hit Enter*
Point to divide-interpolate from?
Pick point or point number: *1*

```
PointNo. Northing(Y) Easting(X) Elev(Z) Description
1         4252.76     4158.32    0.00
```

Point to divide-interpolate to?
Pick point or point number: *pick a point*
Number of Segments-Divisions: *3*
Enter Point Description <>:*hit Enter*
The command then locates two points.

Keyboard Command: divlin
Prerequisite: 2 points

Function

This command locates points along an entity such as a line, polyline, spline or arc. You must specify the number of divisions.

Prompts

Interpolate Elevations [Yes/<No>]: *press Enter*
Select Entity to Divide: *pick point on entity*
Number of Divisions/Segments: *15*
The command then locates 14 points.

Keyboard Command: divent
Prerequisite: 2 points if you want to interpolate elevations

Function

This command creates points at a specified distance along an entity such as a line, arc, spline or polyline. The points are listed out on the text screen, stored in the current coordinate (.CRD) file and drawn on the screen. For example, you might use this command to locate lot corner points along a frontage line. When Break Entity at Points is checked, the selected entity will be broken at every located point. When Create Point at Endpoint is checked,

points will also be located at the endpoints of the selected entity. Horizontal Distance Between Points allow you to specify the distance between located points. There is also an option to create points on curved portions of the centerline at a different interval than on tangent portions (to reduce chord lengths, a shorter interval may be suitable for curves).

For improved descriptions on the points, there is an option, in this main dialog, allowing you to determine whether or not to label elevations on the new points. And for the purposes of describing the points, there is an option that allows you to set the same description to all of the points. For more options related to points, see *Point Defaults*.

Prompts

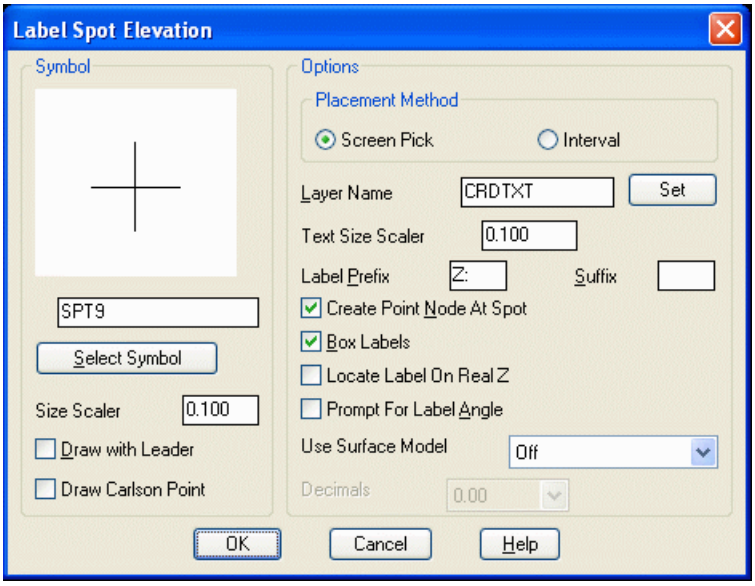
Select entity near endpoint which defines first station.
[nea on] Select Entity to Interpolate Points: *select entity*
[nea on] Select Entity to Interpolate Points: Locating 13 Points
The command locates points along the selected entity.

Keyboard Command: ptint
Prerequisite: An entity

Spot_Elevation

Function

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts

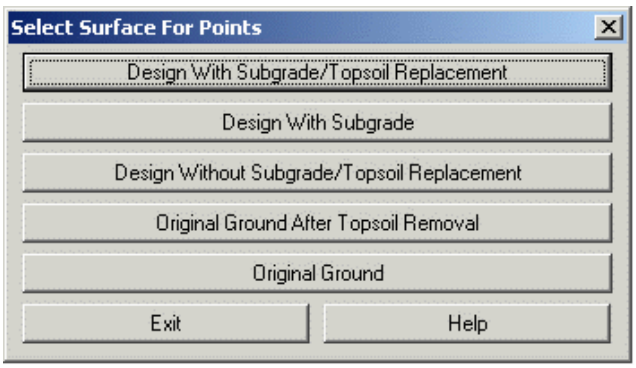
Label Spot Elevation Dialog
Specify a layer name, label prefix and suffix and select the spot symbol.
Point to Label ?
Pick point or point number: *2* Enter a point number.
PointNo. Northing(Y) Easting(X) Elev(Z) Description
2 1231.16 1099.17 30.00 bb
Note: If the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.
Elevation <30.000>: *press enter*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *pick a point on the drawing*
Elevation <0.000>: *enter elevation*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *press enter to finish*

Keyboard Command: LABSPOT
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Create Points On Surface

Function

This command allows you to create points on a selected surface.



Select the surface that you want to create points on, either Design With Subgrade/Topsoil Replacement, Design With Subgrade, Design Without Subgrade/Topsoil Replacement, Original Ground After Topsoil Removal, and Original Ground. This command will calculate the z coordinate of any point that falls within the surface model.

Points can created at various user-specified points or at a specific interval. For individual spot elevations, the user picks or enters the x,y coordinates for each spot elevations. The elevation at the current position of the crosshairs is displayed in real-time in a small window. For interval spot elevations, the alignment for the intervals is defined by a polyline. Then you will be prompted for the interval along the polyline, the number of left offsets and unit interval, and the number of right offsets and unit interval.

Prompts

Command: spotelv

Layer for points <POINTS>:

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Starting Point Number <1>:

Pick spot elevations or interval along polyline [<Pick>/Interval]? i

Pick the centerline polyline:
Enter interval along polyline <50.0>:

Enter number of left offsets <0>: 4

Enter left offset interval <50.0>: 20

Enter number of right offsets <4>:

Enter right offset interval <20.0>:

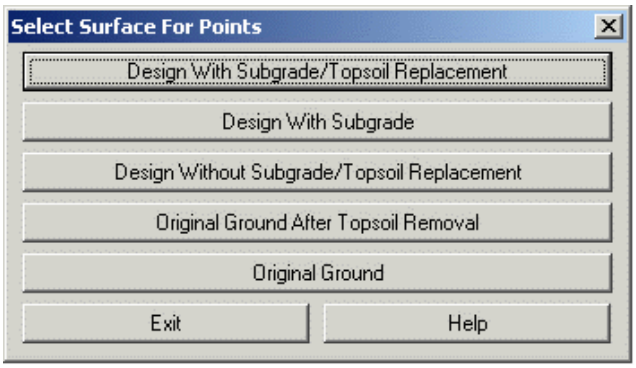
Prerequisite: a surface

Keyboard Command: spotelv

Set Point Elevations To Surface

Function

This command allows you to set point elevations on a selected surface.



Select the surface that you want elevations to be added, either from Design With Subgrade/Topsoil Replacement, Design With Subgrade, Design Without Subgrade/Topsoil Replacement, Original Ground After Topsoil Removal, and Original Ground. Next, pick the points to convert and this command will set the elevations of the points to the surface.

Prompts

Command:
3DCONVERT
Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Select points to convert.
Select objects: Specify opposite corner: 86 found, 25 groups
11 were filtered out.

Select objects:

Converting points...
Converted 25 points.

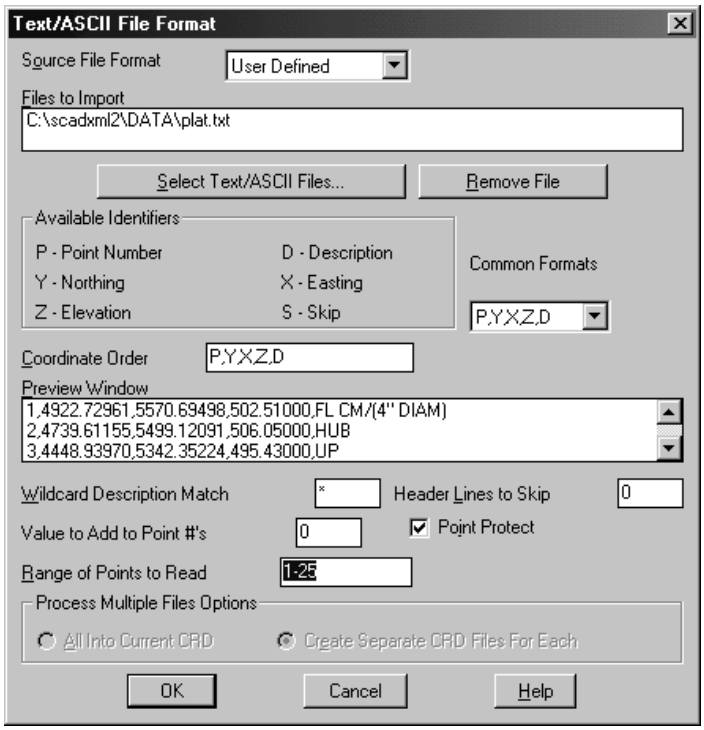
Prerequisite: a surface with an elevation and points to convert

Keyboard Command: 3DCONVERT

Import Text/ASCII File

Function

The Import Text/ASCII File dialog box allows you to convert point data from an ASCII text file into the current coordinate file (.CRD). Each line of the text file can contain any combination of point number, northing, easting, elevation, and description. All of the information for a point should be on one line with the values separated by a comma, space, or other delimiter.



• **Source File Format**: Using this setting, you can choose from specific formats or User-Defined. The special formats of Leica ∗.gsi files, TDS ∗.cr5 files, Geodimeter ∗.obs/∗.raw files, Laser Atlanta ∗.txt files, Trimble ∗.pos files, Zeiss ∗.txt files, Traverse PC ∗.trv files, Maptech, Benchmark ∗.dat files, and Cavantage ∗.cog files, can be directly imported by choosing the corresponding Source File Format.

• **Select Text/ASCII Files**: Using this command, you can select files to read into a coordinate file. Multiple files can be selected by using the Shift or Ctrl keys while picking files. You can also run the Select Text/ASCII Files command multiple times. The files to import are listed in the top scroll display window. The selected file is displayed in the Preview Window to help you fill out the Coordinate Order.

• **Remove File**: This allows you to remove a previously selected ASCII file from the list.

• **Common Formats**: This is a listing of common formats. All the lines in the text file should contain only point data. Any header lines should be removed.

• **Coordinate Order**: For User-Defined, the format of the text file is specified in the Coordinate Order field where the value identifiers are listed with the appropriate delimiters. For example:

1 For a text file with northing, easting, elevation, and comma delimiters:

5100.0,5150.5,485.1

5127.1,5190.3,487.3

The Coordinate Order would be:

Y,X,Z

2 For a text file with point number, easting, northing, elevation, description, and space delimiters:

1 5000.0 5000.0 490.3 TRAV

2 5030.4 4930.5 495.5 TRAV

The Coordinate Order would be:

P X Y Z D

3 When the Coordinate Order is set, click OK to read the text file.

- **Wild Card Descriptions Match**: This allows for only points with matching descriptions to be imported.
- **Header Lines to Skip**: This is an easy way to skip a header in the file to be read. Enter the number of lines not to be processed at the beginning of the text file.
- **Value to Add to Point #'s**: This option allows you to renumber the points as they are imported.
- **Point Protect**: With this option active, the command will check if a point number already exists in the coordinate file before importing the point. If a point conflict is found, you can either assign a new point number or overwrite the old point.
- **Process Multiple Files Options**: The point data from all the imported files can be stored to the current coordinate file or to separate files for each import file. The **All Into Current CRD** option allows you to import one or multiple ASCII files into the current coordinate file. The **Create Separate CRD Files for Each** option will name the coordinate files by the imported file name with a .CRD file extension. For example, the import of file job125.txt would create job125.crd.

Prerequisite: A text file to read.

Keyboard Command: READPT

Import TDS RD5 File

Function

This command converts a TDS RD5 road file into TakeOff centerline (.CL) and profile (.PRO) files.

Prerequisite: a TDS RD5 file

Keyboard Command: importrd5

Import LDD MDB File

Function

This command converts a LDD MDB file into TakeOff centerline (.CL) and profile (.PRO) files.

Prerequisite: a LDD MDB file

Keyboard Command: ldd_crd

Export Text/ASCII File

Function

This command outputs point data from the current TakeOff coordinate file to an ASCII text file. Specify the type of file to write with the Coordinate Order radio buttons. There are several variations on point number, northing, easting, elevation and descriptions as well as specific formats for Leica, Geodimeter, Zeiss, Maptech and D45 formats.

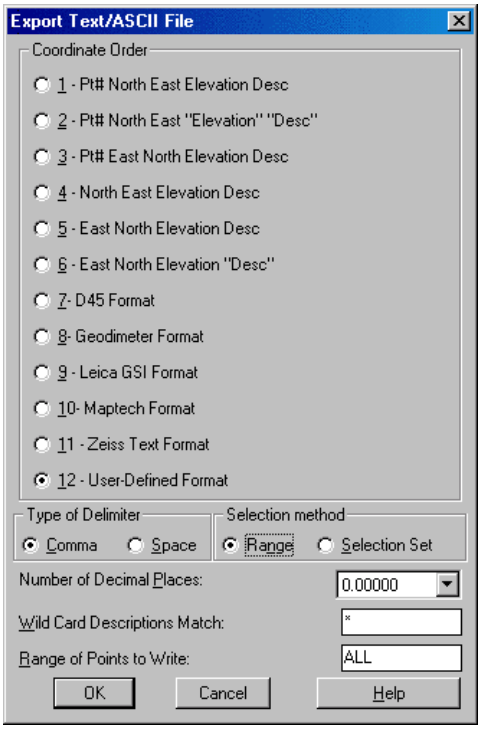
In addition there is an option, User-Defined Format, to define the order of the fields output. When using the User-Defined format, after selecting OK, the User-Define Export Format dialog will appear.

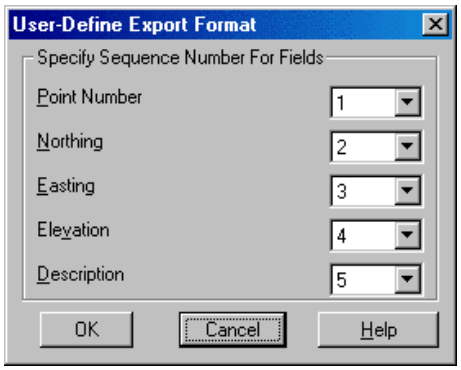
On this dialog, specify the order of the fields by defining a number sequence in each field. You can skip fields and omit data in the output file by leaving None in the sequence field for this data.

The point data to export can be specified by selecting a range of points, or by a selection set of points from the screen. The desired option is specified by selecting the option under the Selection Method on the Export Text/ASCII File dialog. A description filter is also available for exporting only points from the range or selection set with certain descriptions. After selecting the OK button, another dialog appears that allows you to specify a new text?ASCII file or to append data into an existing file. The standard file selection dialog allows you to specify the export file name.

Prerequisite: A Coordinate File (.CRD)

Keyboard Command: writep





Convert Points to AECC POINTs

Function

This command converts Carlson TakeOff points to the AECC_POINT format used by Autodesk Land Development Desktop (LDD). Requires successful installation of the proper Autodesk Object Enabler. This component can be downloaded from the Autodesk web site at http://pointa.autodesk.com/

Prompts

Loading AecBase...

Loading AecCivilBase...

1 Convert all or selected points [All/<Selected>]? **A**

Prerequisite: Points on the screen

Keyboard Command: PT_AEC

Convert Points to Softdesk

Function

This command converts the Carlson TakeOff point blocks in the drawing to Softdesk point blocks. These point block formats are similar and conversion only reorders and renames the attributes.

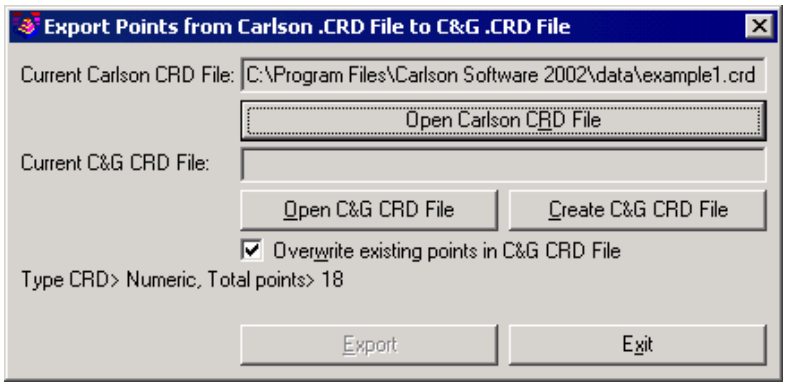
Prerequisite: Carlson TakeOff points

Keyboard Command: 2SOFT

Convert Points to C&G

Function

This command converts a Carlson Takeoff coordinate file into a C&G coordinate file.


Export Points from Carlson .CRD File to C&G .CRD File
Current Carlson CRD File:
C:\Program Files\Carlson Software 2002\data\example1.crd
Open Carlson CRD File
Current C&G CRD File:
Open C&G CRD File
Create C&G CRD File
Overwrite existing points in C&G CRD File
Type CRD> Numeric, Total points> 18
Export
Exit

Start by selecting the Carlson TakeOff coordinate file. After you complete this, the type of coordinate file and total number of points are shown at the bottom. Next choose an existing C&G coordinate file by clicking Open C&G CRD file, or click the Create C&G CRD file button to create a new C&G coordinate file. The last step is to click the Export button.

Prerequisite: Coordinate file

Keyboard Command: CRD2CG

Convert Wild/Leica to Points

Function

This command converts the LisCad or Leica point blocks in the drawing to Carlson TakeOff point blocks. These point block formats are similar and conversion only reorders and renames the attributes. The Update CRD file from Drawing option in the Coordinate File Utilities command can be used directly on LisCad/Leica points without using this command.

Prerequisite: Leica points

Keyboard Command: 2SURV3

Convert Points to Eagle Point

Function

This command converts the Carlson TakeOff point blocks in the drawing to Eagle Point point blocks. These point block formats are similar and conversion only reorders and renames the attributes.

Prerequisite: Carlson TakeOff points

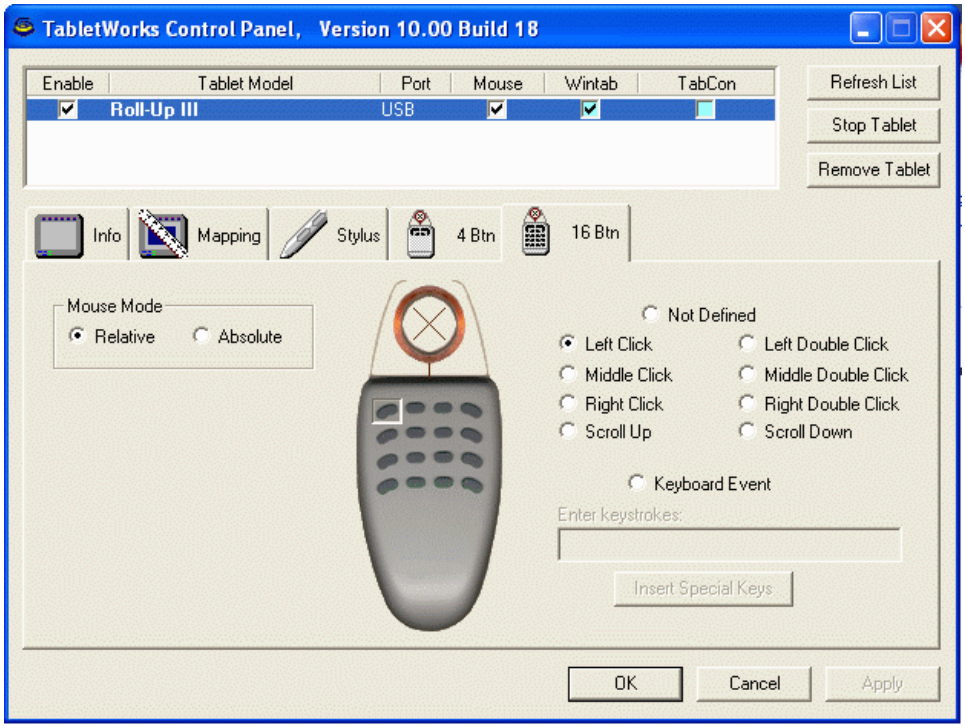
Keyboard Command: 2EDS

Digitize Menu 8

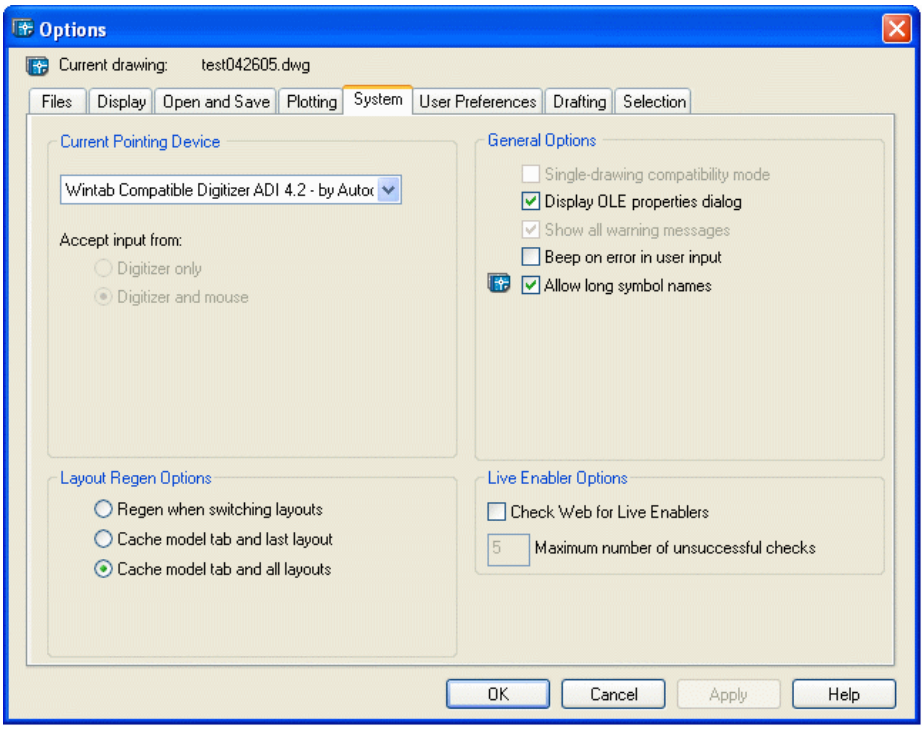
Digitizer Setup

Digitizing is the act of inputting data into the computer by tracing the data from a plan sheet. You need to have a digitizer board, puck, Carlson Takeoff, your computer and your plan to do digitizing. Wintab is a digitizer driver that lets you to use the digitizer cursor as both a digitizer cursor and a mouse. You need to install Wintab when you install Carlson Takeoff. Wintab can be downloaded from GTCO web site: http://www.gtcocalcomp.com/supportgtcosoftware.htm. Select the driver version that suits the type of your digitizer board well.

After you installed Wintab driver on your computer, you set up you digitizer to the correct point mode. In Windows 2000/XP, go to **Start->Settings->Control Panel->TabletWorks**, high light the **16-Btn Cursor**, and select **Mouse** as the **Pointing Mode**, which lets the digitizer cursormoves relatively to the screen coordinates. This step is indicated in the following **TabletWorks Control Panel** dialog.



The next is to set up the pointing device in Carlson Takeoff. Open up Takeoff and go to pull-down **Settings->Preferences**, click tab **System**, select **Wintab Compatible Digitizer** as **Current Pointing Device**, and set the **Accept input from** to **Digitize and mouse**. Please refer to the following **Options** dialog.



Now, you are ready to use your digitizer. On the bottom of the screen, there is a tray icon TABLET on the right side of MODEL. You can use accelerator key **F4** to toggle on/off the tablet.

Tablet On

Function

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.
P.S. Function key [F4] can toggle on/off tablet.
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Keyboard Command: tablet

Tablet Off

Function

Executes AutoCad's TABLET command to set the tablet on. Refer to the AutoCad Reference manual for further information.

P.S. Function key [F4] can toggle on/off tablet.
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Keyboard Command: tablet

Tablet Calibrate

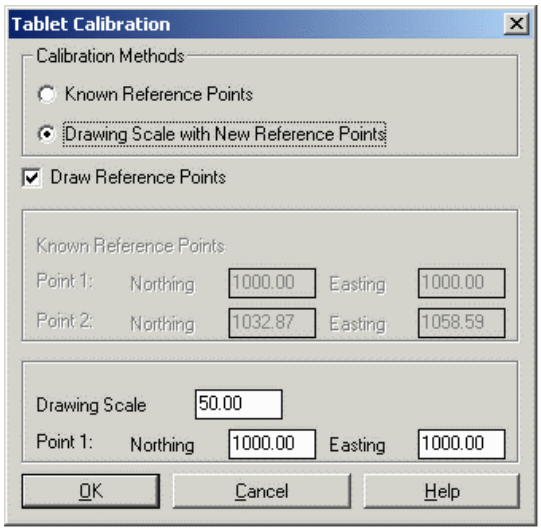
Function

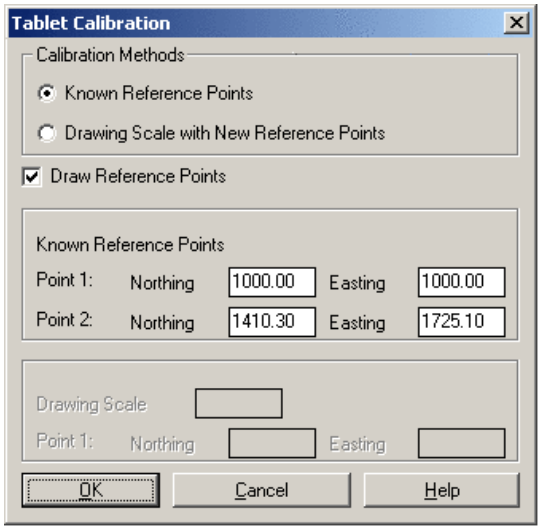
You can calibrate the tablet/digitizer in one of two ways: **Known Reference Points** or **Drawing Scale with New Reference Points**. Reference points are the foundations of whatever data you digitize into the computer. Takeoff bases everything from drawing location to drawing scale on the reference points you digitize.

Drawing Scale with New Reference Points method is very convenient when you don't know the precise coordinates of the entities on your drawing. As long as your can obtain the drawing scale from your plan, this method can establish a coordinate system relative to the position of the plan on the digitizer board. In addition to the drawing scale, you are required to enter a random coordinate for the first reference point, the default coordinate is (1000,1000). Takeoff would computer the coordinate of the second reference point that you pick based on the first point. The coordinates of these two reference points would be saved and will be display on the **Tablet Calibration Dialog** next time when you calibrate the tablet, so you can digitize back to the previous coordinates using **Know Reference Points** method if you are working on the same drawing, though you might have moved or rotated your drawing on the digitize board..

If you know the precise coordinates of two points, you can select **Known Reference Points** method, which establishes a coordinate system that is exactly match the coordinates in the field or on your drawing. Furthermore, Takeoff saves the coordinates of the two reference points from previous calibration and displays them on the **Tablet Calibration Dialog** next time when you calibrate the tablet. If you want to continue to work on the same drawing, you can use the **Know Reference Points** method with the saved coordinates to digitize back to your previous coordinates although you might have moved or rotated your drawing on the digitizer board.

For accurate takeoff calculations, choose two points that can be easily found in the field and are farther apart rather than closer together.





Prompts

Tablet Calibration Dialog
Specify the Calibration Methods. If you select Drawing Scale method, enter the drawing scale and the coordinate of the first reference point. Otherwise enter the exact coordinates of the first and second reference points.
Pick first reference point:*pick a point on the drawing*
Pick second reference point:*pick another point on the drawing*
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up.

Keyboard Command: digsetup

Save Tablet Calibration

Function

This command saves current tablet calibration to a file. You are prompted to enter a file name.

Prerequisite:
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: tablet1

Load Tablet Calibration

Function

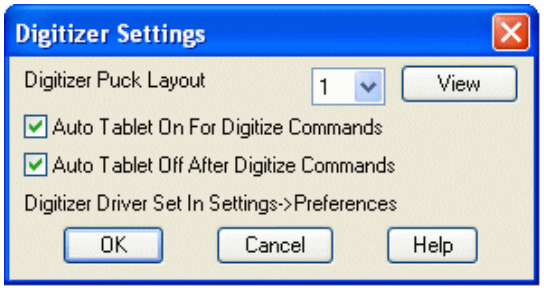
This command restores the tablet calibration parameters from a file and load it into the current drawing. You are prompted to specify a file name.

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. The calibration file should be associated to the current drawing, and the current drawing shouldn't have been moved on the digitizer board since last calibration.

Keyboard Command: tablet2

Digitizer Settings

This command allows you to select the puck layout and set Auto On/Off features.



Auto Tablet On For Digitize Commands means after you select a digitize command your puck will automatically be put in Digitize Mode. If this is toggle off, then you will need to turn Tablet on seperately from running a digitize command.

Auto Tablet Off After Digitize Commands means you will return to Mouse Mode after running a digitize command. Read below for more on Mouse and Digitize Mode.

Puck Layout

The 16-button puck can be used as either a mouse or a digitizer. It's very important to understand how the 16 buttons are mapped in both modes.

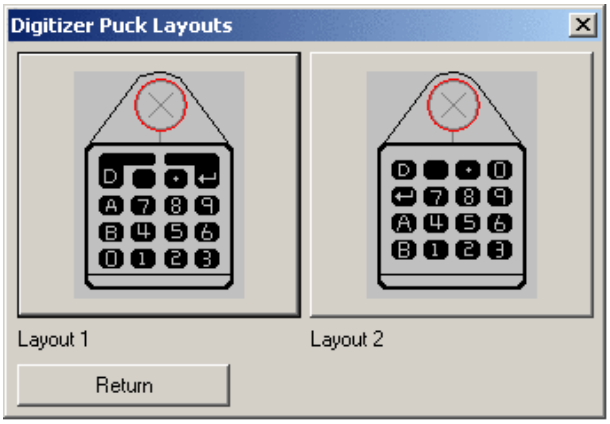
Mouse Mode:

When the tablet is off, the puck is in Mouse Mode. The top-left button is the left mouse click, and the top-right button is the right mouse click. The labels on the other buttons do not mean anything. All buttons are mapped as same as the buttons of the default pointing device in AutoCad . Please refer to AutoCad Reference manual for further information.

Digitize Mode:

When tablet has been calibrated and is on, the puck is in digitize mode. It is mapped as a small keyboard, which enables you to enter numerous values such as elevation, thickness and offset etc., and also provide you some functionality to digitize various entities. Currently there are two puck layouts in Takeoff, shown in the figure below. After you install Carlson Takeoff and finish setting up the digitizer, you go to the pull-down menu **Digitize->Puck Layout** to select a 16-button puck layout. A button mapping would be created andTakeoff would recognize the buttons as represented.





Layout 1 is Carlson Puck Layout, which is the most common layout used in Carlson Takeoff. Layout 2 is for users who don't have a Carlson Puck. If your puck is different than these two layouts, please contact Technical Support for help setting the mapping for your 16 button puck.

Prompts

Digitizer Settings Dialog
Specify the Digitizer Puck Layout to layout 1 or 2

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed.
Keyboard Command: dig_config

Existing

Function

This sets the layer target to existing. Set this prior to running any digitizing command and anything you digitize will be assigned for your existing surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Prerequisite: none

Keyboard Command: set_digit_exist

Design

Function

This sets the layer target to design. Set this prior to running any digitizing command and anything you digitize will be assigned for your design surface. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

Prerequisite: none

Keyboard Command: set_digit_final

Other

Function

This sets the layer target to other. Set this prior to running any digitizing command and anything you digitize will be assigned to the Other target. Checkout the Define Layer Target/Material/Subgrade command under Tools for more on targets.

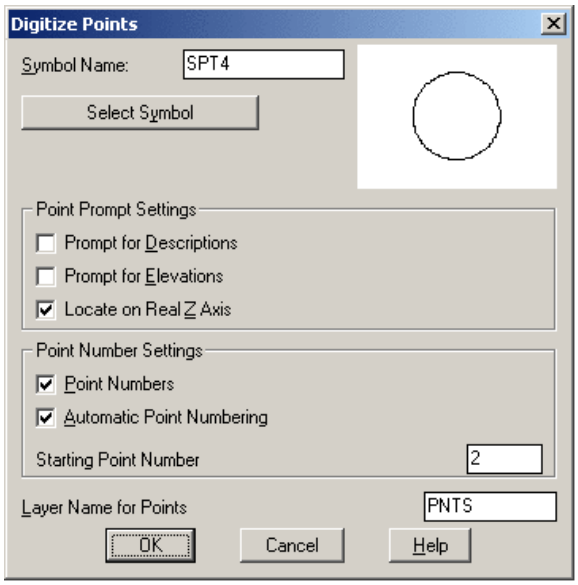
Prerequisite: none

Keyboard Command: set_digit_other

Point

Function

This command allows you to digitize individual points one at a time. The first time it prompts you the **Digitize Points Dialog** for entering point symbol styles, point prompt settings and number settings,starting point number and layer name. If you want to enter the elevation and description for each point, select **Prompt for Descriptions** and **Prompt for Elevations**. Afterhaving digitized a point, you can continue to digitize next point by picking the point on the drawing. The command defaults to the last layer name, point symbol, elevation, description and the last point number plus 1. If you have finished digitizing points, press **Enter** to finish.



Prompts

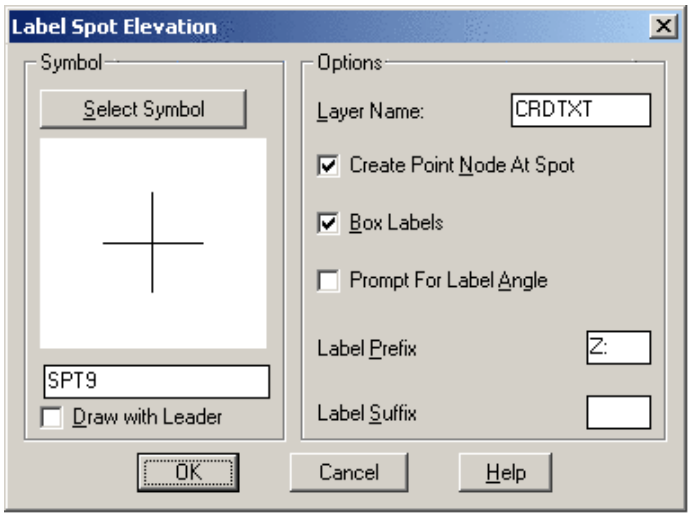
Digitize Points Dialog
Specify a layer name and select the point symbol, point prompt settings and number settings.
Pick point to create (Enter to end): *pick a point on the drawing*
Select/<Enter Point Elevation <>>:*enter the elevation or type <Select> to select the elevation text on the screen*
Enter Point Description <>:*enter the point description*
Result like "N: 1231.16 E: 1099.17 Z: 30.00" would be display on the command line, and a point would be drawn on the screen with the text of its number, elevation and description.
Pick point to create (Enter to end):*pick next point or press Enter to finish digitizing points*
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: dig_pt

Spot Elevation

Function

This command allows you to label points with their elevation. The point can either be digitized from a drawing, picked on a screen or specified by a point number. The command first prompts you the **Label Spot Elevation Dialog** for entering layer name, label prefix and suffix and symbol types etc. Click OK to start. After specifying the point, the command prompts you to enter the elevation if its elevation is unknown and then pick an angle from the location of the point to label the elevation. You can repeat labeling points until you press **Enter** to finish.



Prompts

<>**Label Spot Elevation Dialog**
Specify a layer name, label prefix and suffix and select the spot symbol.
Point to Label ?
Pick point or point number: 2 *(enter a point number)*
PointNo. Northing(Y) Easting(X) Elev(Z) Description
2 1231.16 1099.17 30.00 bb
Note: if the point number you entered is not in the drawing, you will be prompted again to pick point or enter a point number.
Elevation <30.000>: *press enter*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *pick a point on the drawing*
Elevation <0.000>: *enter elevation*
Pick angle for label: *pick an angle from the spot*
Point to Label (ENTER to End)?
Pick point or point number: *press enter to finish*

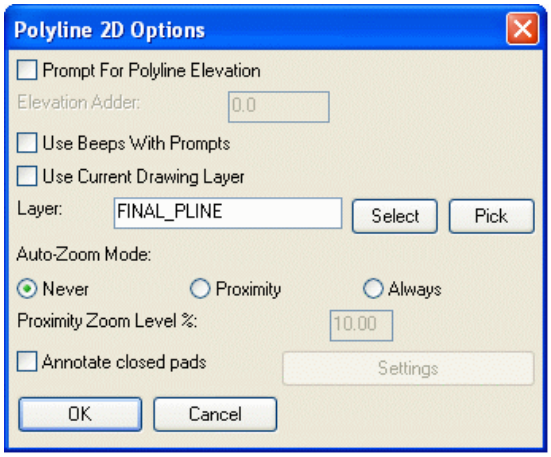
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: labspot

2D Polyline

Function

A 2D polyline is a line of connected points that have the same elevation. This command lets you digitize a 2D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 2D Options Dialog** for entering the layer name. **Prompt For Polyline Elevation** option allows you to enter the elevation for each polyline, otherwise all 2D polylines have 0.0 elevation. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you getnear the edge of the screen while picking points. There are three ways to enter a layer name, **Use current drawing layer**, **Select** from a list of layername, or **Pick** an entity on the screen to get its layer name. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself . If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts

Polyline 2D Options Dialog
Enter the layer name and select the options of Prompt For Polyline Elevation and Auto-Zoom mode etc.
Enter default elevation <0.00>: *100*
First point: *pick a point on the drawing using puck*
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 119.03, Total length: 119.03
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*

Segment length: 121.76, Total length: 240.80
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 115.23, Total length: 356.03
Close[A]/Undo[B]/Pick next point (Enter to end): *press enter to finish digitizing or press A to close the polyline*
Digitize Another FINAL Polyline [Yes(A)/<No(B)>]? *press A on the puck or enter Yes on the keyboard to digitize next 2D polyline, press B on the puck orenter No on the keyboard to finish digitizing 2D polyline.*

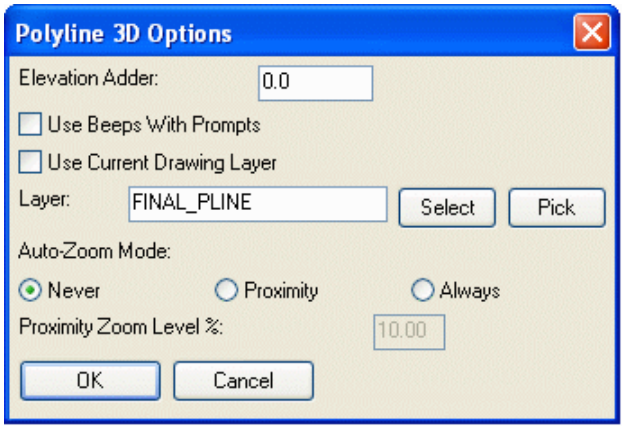
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: dig_2dp

3D Polyline

Function

A 3D polyline is a line of connected points that have various elevations, and the slope between points is constant. It can be used in defining pads, excavations, drainage ditched and slopes from proposed design features to meet existing site conditions. This command lets you digitize a 3D polyline by picking points along the lines on the drawing. It prompts you first the **Polyline 3D Options Dialog** for entering the layer name. **Elevation Adder** allows you to truncate the elevations you have to enter in by add a given amount tothem. There are five ways to enter elevations: known elevation of the point, interpolate, slope from previous point, ratio from previous point and degree from previous point. You can choose one of the methods between picking points. **Auto-Zoom** mode would automatically zoom the display to center around the last point when you getnear the edge of the screen while picking points. While digitizing a polyline, press A to interpolate the elevation or B to enter it in. The command keeps prompting you to pick the next point until your **press Enter** to finish digitizing, or **press A** on the puck or **enter Close** on the keyboard to close the polyline on itself. You can also use AutoCAD's OSNAP command to pick points by pressing the decimal [.] button on the digitizer puck. If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a polyline, the command prompts your to digitize another polyline until you **press B** or **enter No**.



Prompts

First point:
Interpolate[A]/screen Pick/<Elevation[B]> <0.00>: 256

Z: 256.00
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point
Slope/Ratio/Interpolate[A]/Degree/screen Pick/<Elevation[B]> <256.00>: A
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Pick point
This point elevation will be interpolated upon completion.
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: 279

Z: 279.00, Hz dist: 30.01, Slope dist: 37.81, Slope: 76.6% Ratio: 1.3:1
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): Pick point
Slope/Ratio/Elevation[B]/Degree/screen Pick/Osnap[.]/Next point or elevation<Interpolate>: Press Enter

Z: 279.00, Hz dist: 24.18, Slope dist: 24.18, Slope: 0.0% Ratio: 0.0:1
Close[A]/Undo[B]/Osnap[.]/Pick next point (Enter to end): A

Digitize Another EXIST_PLINE Polyline [Yes(A)/<No(B)>]? B
<**Tablet Off**>

Prerequisite:
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command:
dig_3dp

Perimeter

Function

Perimeter is a 2D polyline that all points on it have the same elevation. It can be used as boundary polyline of your targets on your drawing. This command allows you to digitize a perimeter by picking points on the drawing. While digitizing a polyline, the command keeps prompting you to pick next point until your **press Enter** to finish digitizing, or **press A** on the puck or**enter Close** on the keyboard to close the polyline on itself . If you make a mistake, **press B** on the puck or **enter Undo** on the keyboard to remove the mistake and then continue to digitize. After finishing a perimeter, the command prompts your to digitize another polyline until you **press B** or **enter No**.

Prompts

First point: *pick a point on the drawing using puck*

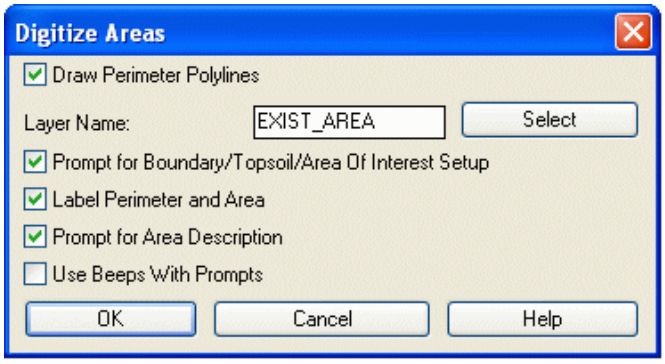
Segment length: 0.00, Total length: 0.00
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 104.27, Total length: 104.27
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 153.14, Total length: 257.41
Close[A]/Undo[B]/Pick next point (Enter to end): *pick next point*
Segment length: 104.89, Total length: 362.30
Close[A]/Undo[B]/Pick next point (Enter to end): *press Enter to finish the perimeter, or press A to close the perimeter*
Digitize Another PERIMETER Polyline [Yes(A)/<No(B)>]? *press A or enter Yes to continue digitizing another perimeter, press B or enter No to finish digitizingperimeters.*
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: dig_perim

Areas

Function

This command allows you to find an area in digitize mode. With the puck, pick around the area you wish to calculate. If Draw Perimeter Polyline is toggled on then the linework of your perimeter will be displayed. You can then set the Layer Name and choose to label the Perimeter and Area and enter in an Area Description. You can also set the area you created as a Boundary, Topsoil, or Area of Interest.



Prerequisite: a digitizer

Keyboard Command: dig_area

Contour Polyline

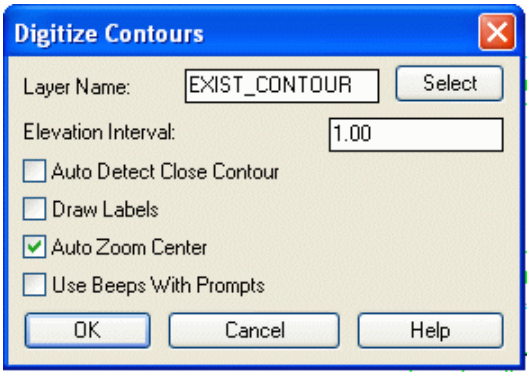
A contour is a line of points with a constant elevation, representing the natural contour of the site. In Takeoff, there are two layer targets: Existing Ground Surface and Design Surface. Contour Polyline has two sub-command to digitize contour lines into Existing Contour and Final Contour layers directly for assigning them easily into Existing Ground Surface and Design Surface in the future analysis.

There are two ways to digitize contour lines: sketch mode or point mode. You can start digitizing a contour with one mode and switch to the other during digitizing the contour. Sketch mode uses more points than pick mode. In general, we recommend using pick mode to digitize the straight parts of lines because it reduces the number of points and speeds up Takeoff's calculations, but using sketch mode to digitize the curved parts because it is fast and accurate.

Function

This command lets you digitize contours as polylines one at a time. The first time it prompts you the Digitize Contours Dialog. Enter the layer name or select it from a list of existing layer. Look at your plans and determine an elevation interval that is between most of the contours and enter it in the Elevation Interval field. You are able to modify both the value and the direction of the elevation interval between digitizing contour lines, using the buttons on the puck. To have Takeoff automatically close contours whose beginning and ending points are within a specified range, check the Auto Detect Close Contour. Draw Labels would draw the elevation at the starting point of the contour. In Pick mode, if you want the Takeoff to automatically zoom the display to center around the last point when you get near the edge of the screen while picking points, check the Auto Zoom Center. Click OK to start digitizing.

If this is your first time digitizing a contour, you are defaulted to the Pick Mode digitizing, otherwise you would be defaulted to the last digitize mode. If you want to use the other digitize mode, press 0 on the puck or enter 0 from the keyboard. Place your cursor at one end of the contour line and begin digitizing the line. While digitizing a line, you can force a contour to close on itself by pressing A on the puck to end the contour and connect the last point to the first point, remove a mistake by pressing B on the puck, or switch to the other digitize mode by pressing 0. During Sketch Mode digitizing, you can stop digitizing by pressing Pick or Enter button on the puck, take some rest or changes, and start sketching again. At the end of the contour line, **press Enter** on your puck or keyboard. The contour is completed, and the elevation for the next contour is automatically incremented. You would be asked to digitize next contour. If you press A on the puck or enter Yes on the keyboard, you can digitize another contour, or press B on the puck or enter No on the keyboard to finish digitizing contours.



Prompts

Digitize Contours Dialog
Enter Layer Name, Elevation Interval, and toggle on/off Auto Detect Close Contour etc.
Increment(1.00)[A]/Direction(+)[B]/Elevation <573.00>: 450 *(enter elevation or press Enter to accept current value)*

Start Digitizing...

Sketch[0]/Pick the first point: *pick a point to start Pick Mode digitizing (press 0 to switch to Sketch Mode)*
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): 0(press 0 on the puck or enter 0 on the keyboard to use Sketch Mode)

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): B (undo the last point)
Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): pick and drag again
Drag to digitize (Pick or press Enter to stop sketching)... pick or press Enter to stop sketching

Pick[0]/Close[A]/Undo[B]/Pick and drag (Enter to end): 0 *(press 0 on the puck or enter 0 on the keyboard to use Pick Mode)*

Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): pick next point
Sketch[0]/Close[A]/Undo[B]/Pick next point (Enter to end): press Enter to finish digitizing

Digitize Another Contour [<Yes(A)>/No(B)]? B(press B to finish digitizing)

Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.

Keyboard Command: digcont_exist, digcont_final

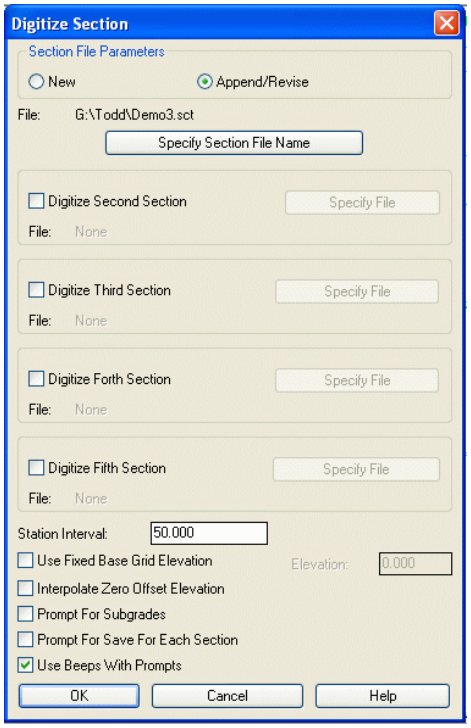
Sections

Function

This command allows you to digitize section lines and store the section data in the section file you have specified. The command first prompts you the **Digitize Section Dialog**. Enter the section file name and determine if you want to digitize second and third sections at the same station. Look at your plans and determines the station interval, which is used to automatically default to the next station value when digitizing a series of stations. If the grids at all the stations have the same base elevation, toggle on Use Fixed Base Grid Elevation. You can also toggle on

Interpolate Zero Offset Elevation, Prompt for Subgrades, Prompt for Save for Each Section and Use Beeps with Prompts. Click OK to start digitizing.

Takeoff prompts you to calibrate the section sheet before you digitize the section lines. You pick three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals. Corners on the section grid are preferred reference points. Place your cursor at one end of the section line and begin digitizing the line. While digitizing a line, you can remove a mistake by **pressing A** on the puck or **entering Undo** on the keyboard. At the end of the section line, **press Enter** on your puck or keyboard. The station is completed, and the station value isautomatically incremented. The command would prompts to digitize next section. You can **press A** on the puck or **enter Exit** on the keyboard to finish digitizing. If you want to continue to digitize next section, **press Enter** or enter the new station number. For every station after the first one, you can calibrate the grid sheet by picking one reference point and specify its offset and elevation. After you digitize the section lines on your drawing, all the section data would be saved in a section file (.sct).



Prompts

Digitize Section Dialog

Enter Section File Name, Station Interval, and toggle on/off Use Fixed Base Grid Elevation etc.

Section station to digitize <0.000>: *press Enter to start with station 0.0 or enter a station number*

Calibrate section sheet

Pick First section sheet reference point: *pick a grid point of this station on your drawing*

Enter offset <0.0>: *press Enter to accept the offset or enter the offset of the point to the centerline*

Enter elevation: 1030 *(enter the Elevation of the reference point)*

Pick Second section reference point: *pick the second grid point*

Enter offset: 0 *(enter the offset of the point to the centerline)*
Enter elevation: 1040 *(enter the Elevation of the reference point)*
Pick Third section reference point: *pick the third grid point*
Enter offset: 50 *(enter the offset of the point to the centerline)*
Enter elevation: 1040 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible
RMS Error: 11.49
Standard deviation: 2.38
Largest residual: 14.08
At point: 2
Second-largest residual: 14.08
At point: 1

Digitize break point for DRAWING1 section 0.000 (Enter to end): *pick a point on the section line*
Offset: -39.81 Elev: 1028.80
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: -9.94 Elev: 1030.03
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 49.44 Elev: 1034.93
Digitize break point for DRAWING1 section 0.000 (Undo[A],Enter to end): *press Enter to finish*
Save changes to DRAWING1 section 0.000 [<Yes(A)>/No(B)]? A *(press A or B)*
Exit[A]/Section station to digitize <50.000>: 200 *(enter next station number)*
Calibrate next section
Pick section reference point: *pick a grid point of the station on your drawing*
Enter offset <0.00>: *press Enter to accept the offset or enter the offset of the point to the centerline*
Enter elevation <1030.00>: 1020 *(enter the Elevation of the reference point)*
Digitize break point for DRAWING1 section 200.000 (Enter to end): *pick a point on the section line*
Offset: -40.40 Elev: 1008.07
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: -5.38 Elev: 1019.98
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 27.86 Elev: 1030.02
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *pick a point on the section line*
Offset: 50.33 Elev: 1035.80
Digitize break point for DRAWING1 section 200.000 (Undo[A],Enter to end): *press Enter to finish*
Save changes to DRAWING1 section 200.000 [<Yes(A)>/No(B)]? A *(press A or B)*
Exit[A]/Section station to digitize <250.000>: A *(press A to finish or enter the station number to continue)*
Prerequisite: Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.
Keyboard Command: digxsec

End Areas

Function

There are two types of end areas: cut area and fill area. This command allows you to digitize both cut area and fill area on the drawing and writes data to a .ew file. The command first prompts you to calibrate the section sheet by picking three points and specify their offsets to the centerline and elevations in order to determine the horizontal and vertical intervals .Corners on the section grid are preferred reference points. Then it prompts you to digitize the cut area and fill area respectively. Place your cursor at one end of the end area and begin digitizing the outline of the area. At the end of the section line, **press Enter** on your puck or keyboard. The end area is completed, and its area is printed on the command line, and you are prompted to digitize next end area. After you finish all the end area at one station, accumulated cut area and fill area are computed and printed out on the screen. All data of cut area and fill area at every station would be saved in the area file (.ew) that you have specified.

Prompts

Calibrate section sheet
Pick First section sheet reference point: *pick a point on the drawing*
Enter offset <0.0>: *press Enter to accept the offset (or enter the offset of the point to the centerline)*
Enter elevation: 1020 *(enter the Elevation of the reference point)*
Pick Second section reference point: *pick a point*
Enter offset: 0 *(enter the offset of the point to the centerline)*
Enter elevation: 1030 *(enter the Elevation of the reference point)*
Pick Third section reference point: *pick a point*
Enter offset: 50 *(enter the offset of the point to the centerline)*
Enter elevation: 1030 *(enter the Elevation of the reference point)*

3 calibration points

Transformation type: Orthogonal Affine Projective

Outcome of fit: Success Exact Impossible
RMS Error: 11.69
Standard deviation: 2.40
Largest residual: 14.29
At point: 2
Second-largest residual: 14.29
At point: 3

Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area,* 0*(0.211129 1030.76)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area,* 1*(11.5804 1030.49)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area,* 2*(17.8643 1030.73)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area,* 3*(19.0216 1032.35)
Digitize cut area (Enter to end): *pick a point that is on the outline of the cut area,* 4*(-0.777246 1030.75)
Digitize cut area (Enter to end): *press Enter to finish*
End area: 17.2312
Accumulated Cut Area: 17.2312
More Cut Areas [Yes(A)/<No>(B)]? *press A to digitize more Cut Areas, or press B to finish digitizing Cut Areas.*

Accumulated Cut Area: 17.2312
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 0*(-18.9614 1029.65)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 1*(-18.1315 1030.75)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 2*(-11.9592 1030.49)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 3*(-2.06761 1030.72)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 4*(-10.0082 1030.01)
Digitize fill area (Enter to end): *pick a point that is on the outline of the fill area,* 5*(-18.531 1029.67)
Digitize fill area (Enter to end): *press enter to finish*
End area: 8.64646
Accumulated Cut Area: 8.64646
More Fill Areas [Yes(A)/<No>(B)]? *press A to digitize more Fill Areas, or press B to finish digitizing Fill Areas.*
Accumulated Cut Area: 8.64646
Total Cut Area: 17.2312
Total Fill Area: 8.64646
Store data to file [<Yes>(A)/No(B)]? *press A or B*

Opened file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew
Station Number: 1 *(enter Station Number)*

Data Stored in file: C:\Program Files\Carlson TakeOff 2004\DATA\Drawing1.ew
Digitize another station [<Yes>(A)/No(B)]? B (*press A or B)*

Prerequisite:
Have a digitizer board and a puck connected to your computer, and have Wintab driver installed. The digitizer has been correctly set up. Have done tablet calibration for current drawing.<>

Keyboard Command: digendar

Elevate Menu 9

Change Elevations

Function

This command will change the elevation of selected Entities. It can move the entity to a specified elevation from it's current elevation (absolute) or do a differential change by adding or subtracting a value from it's current elevation. If Carlson TakeOff points are selected, their attribute text and z axis coordinate are changed.

Prompts

Ignore zero elevations (<Yes>/No)? *Press Enter.* If you answer *No*, then entities with elevation 0 will be changed.

[A]bsolute or [D]ifferential Change <A>: *A*

Select/<Enter Elevation <0.0000>>: *125*

Change Layer for changed entities [Yes/<No>]: No
Elevation to change to:

By using the Absolute option all entities selected are changed to the elevation 125.

Select Entities for elevation change.

Select objects: *C*

First corner: *(pick point)*

Other corner: *(pick point)*

Select objects: *[Enter]*

Prerequisite: Something to change

Keyboard Command: chgelev

Set Polyline to Elevation

Function

This command allows you to assign elevations to one or more polylines. The elevation can be assigned by entering in the value or by picking a text entity that has the elevation.

Prompts

Command: set_pline_z

Select/<Enter Elevation <0.0000>>: Select a text entity or type in an elevation. Press enter for the default elevation in brackets.

Select Polyline for elevation change. Pick on the screen a polyline you wish to change such as:
LWPOLYLINE
Done.
Set another polyline [<Yes>/No]? Press Y to pick another polyline to assign an elevation to. Type in N to finish the command.

Prerequisite: A polyline and an elevation to assign it.

Keyboard Command: set_pline_z

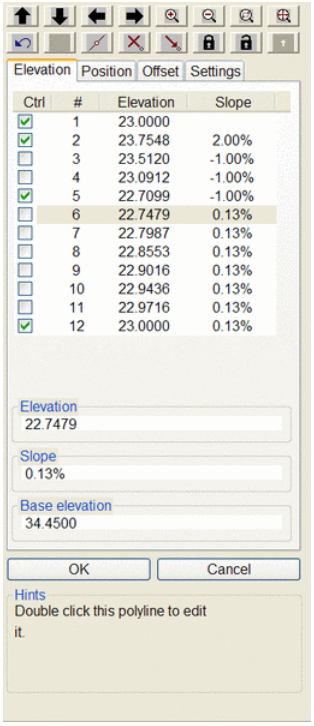
Edit-Assign Polyline to Elevations

Function

This command allows very precise control of 3D polylines, specifically in the ability to edit vertex elevations, as well as add, delete, or move vertices. You can also control the location of polyline vertices as defined by the station and offset of the vertices relative to a Centerline.

Polyline vertices are designated as either Control or Free vertices. The elevation of Control vertices are set and held, the elevations of Free vertices are interpolated. In the drawing, control vertices are shown with red boxes, along with their vertex number and elevation. Free vertices are displayed with blue boxes and are not annotated.

When you run the command, you are first prompted to select a polyline to edit. When you pick a polyline to work with, the following control panel appears on the left side of your screen.



The top row of buttons across the top of the control panel are used to manipulate the view in the drawing with various Zooming and Panning options. The second row of buttons will change as you select different tabs, but are essentially used to add vertices, delete vertices, or pick elevations or locations for vertices.

The four tabs in the panel provide access to control of polyline vertex Elevation, Position, Offset and Setings.

Elevation: This tab displays the vertices of the poyline, each with a check box to set whether it is a control vertex or free, its assigned number, its elevation, and the slope from the previous vertex to that vertex. Selecting a vertex

highlights its grip in the drawing. Once selected, you can enter an elevation or slope for that vertex in the spaces below the list, thereby automatically setting the vertex to a control vertex. The Base Elevation is used to adjust the elevations of all the vertices simultaneously.

Position: The Position tab displays the coordinates of each vertex. To move a vertex, you can type in new coordinates, use the Pick Position icon to specifiy a new location for the vertex on the screen, or you can grip the vertex and drag it to a new location.

Offset: The Offset tab requires the selection of a Centerline to reference. Once a Centerline is designated, the Station, Slope, and Offset of each vertex relative to the Centerline is displayed and can be edited.

Settings: The Settings tab provides control over various overall options pertaining to the use of the command. For example, hinding free vertices and setting how to report your slopes between vertices.

Right-click menu: There is a right-click menu available at all times which also gives access to a variety of functions and settings.

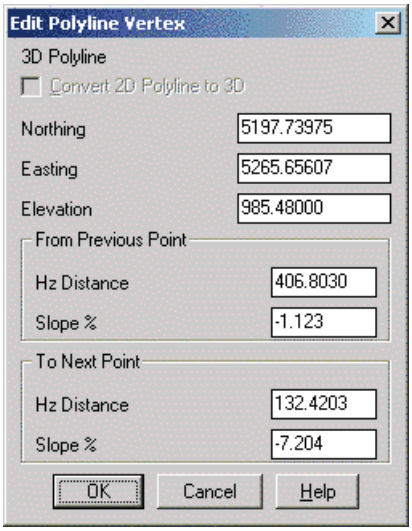
Keyboard Command: edit_pline_z
Prerequisite: Polylines with vertexes

Edit Polyline Vertex

Function

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyine. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.



Prompts

Select polyline vertex to edit: *pick a polyline at the point to be modified*

Pick or enter position <5264.23,5048.21>: *pick a point*

Enter elevation <0.00>: *Press Enter*

Select polyline vertex to edit: *Press Enter to end*

Prerequisite: A polyline.

Keyboard Command: editpl

2D to 3D Polyline by Points

Function

This command converts a 2D polyline into a 3D polyline by using the elevations of points. At each vertex of the polylines, the program looks for a point with elevation at the same x,y location. The points can be Carlson point blocks or AutoCAD POINT entities. This routine can be useful if the linework is created in 2D at zero elevation, and points with elevation are located along the linework. Then the linework can be converted into 3D polylines with this command. For example, a centerline polyline with arcs may need to be created in 2D for stationing because AutoCAD does not allow arcs on 3D polylines. To use this polyline as a breakline in surface modeling, this command can convert the polyline into a 3D polyline.

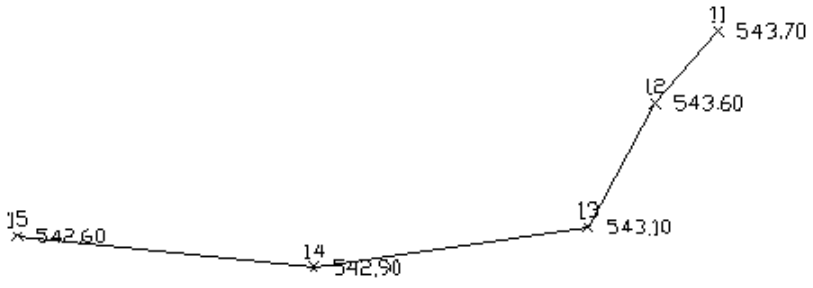
Prompts

Select points and polylines.

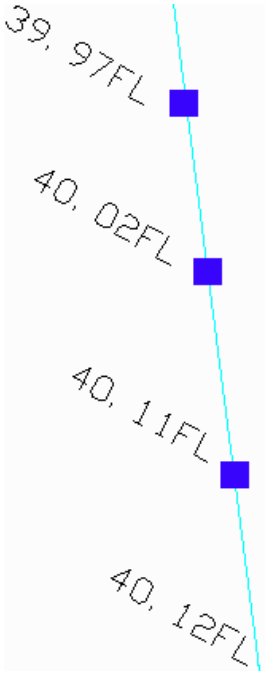
Select objects: select polylines to convert and the points with elevation

Prerequisite: A polyline and points

Keyboard Command: 2dto3dpt



2D to 3D Polyline-By Text



Function

This command allows you to change 2D polylines to 3D polylines by elevation labels.

This command will prompt you for samples of the elevation labels and the polylines to convert. The program uses these samples tp know the layer names for the labels and linework to process. Then select all the polylines with their labels you want to convert.

You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 539.97, 540.02, 540.11 sometimes, like in the example on the side, they are listed as 39.97, 40.02, 40.11. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.
This command will assign elevations from the labels to nearby vertices. If vertices do not have a close elevation label than they will be interopated from vertices that are nearby elevation labels.

Prompts<></>

Command: elevfb

Select sample of elevation text: Pick a text label
Select sample of a polyline to convert: Pick a polyline
Select polylines to convert and elevation labels.

Select objects: Select all the entities to process
19 found, 19 total

Enter elevation to add to label values <0.00>: 500
Pre-processing entity #19 of 19
Filtering text entities
Processing elevation text #18
Remaking polyline #1

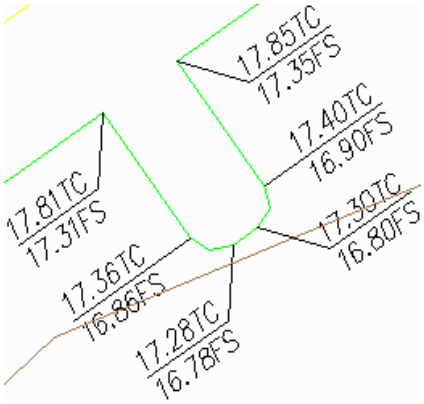
Prerequisite: 2D polyline and elevation labels

Keyboard Command: elevfb

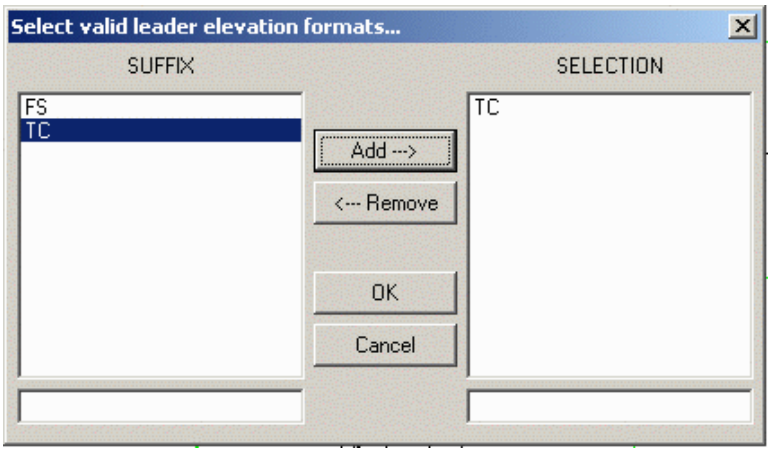
2D to 3D-By Text With Leader

Function

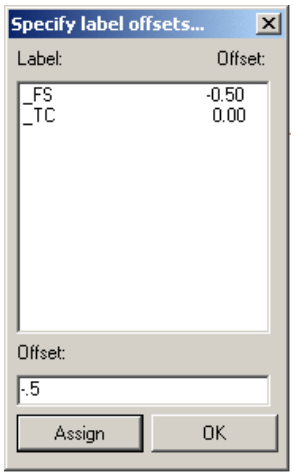
This command will assign elevations from the labels to the polylines by following the label leaders to their corresponding vertices on the polyline.

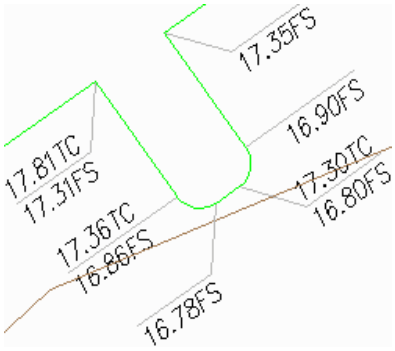


This command will prompt you for samples of the elevation labels, the leaders, and the polylines to convert. The program uses these samples tp know the layer names for the labels and linework to process. Then select all the labels and leaders for the polylines you want to convert. You will then be prompted to enter in an elevation to add to label values. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 800s instead of labeling every elevation 817.85, 817.40, 817.30 sometimes, like in the above example, they are listed as 17.85, 17.40, 17.30. This command allows you to add a given amount, such as 800, toevery label elevation to produce the correct elevation in the drawing.



Carlson TakeOff searches for all leaders and gathers their associated text. If the program finds different labels inthe elevation text, then this dialog box allows you to select the text you want to create 3D polylines. In this example you might want to use elevations followed by TC. This dialog box allows you to select that text and exclude the other text which is not to be used in the elevations of the polyline, such as FS.





If you are creating 3D polylines from multiple elevation labels than this dialog box will allow to offset certain labels by a given amount. In the above example you can offset an elevation labeled FS by .50 so that it matches vertices set by TC labeled elevations.

Prompts

Command: elevfl

Select sample of elevation text: Pick a text label
Select sample of an annotation leader: Pick an annotation leader
Select sample of a polyline to convert: Pick a polyline
Select polylines to convert, leaders and elevation labels to process.
Select objects: Select the desired entities
22 found
3 were filtered out.

Select objects:

Enter elevation to add to label values <0.00>: 800
Pre-processing entity #19 of 19
Filtering text entities
Processing leader #6
Remaking polyline #1

Prerequisite: 2D polyline, elevation labels, and leaders

Keyboard Command: elevfl

2D to 3D Polyline by Start/End Elevations

Function

This command allows you to convert a 2D polyline to a 3D polyline by specifying the starting and ending elevations of the polyline. All intermediate polyline vertice elevations are linearly interpolated from these end point elevations.

Prompts

Select polyline to assign elevations:

Enter starting elevation: *109.85*

Percent/Ratio/<Enter ending elevation>: *112.16*

Select polyline to assign elevations (Enter to End): *press enter to end*

Prerequisite: A polyline

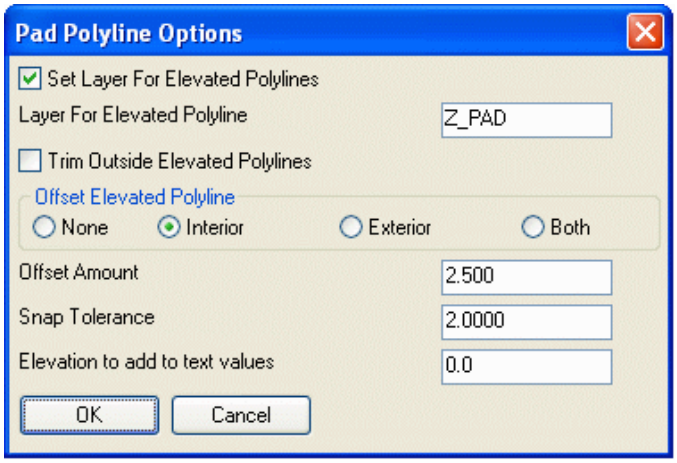
Keyboard Command: 2dto3dpl

Pad Polyline By Interior Text

Function

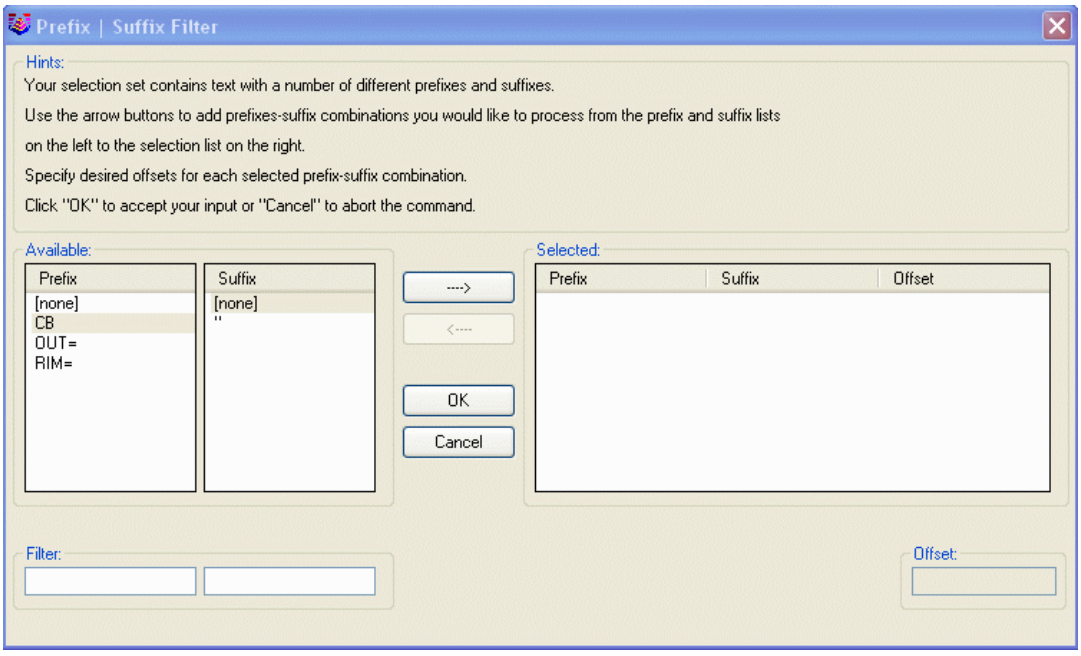
This command allows you to set one or more pad elevations using interior text labels.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundry of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework.

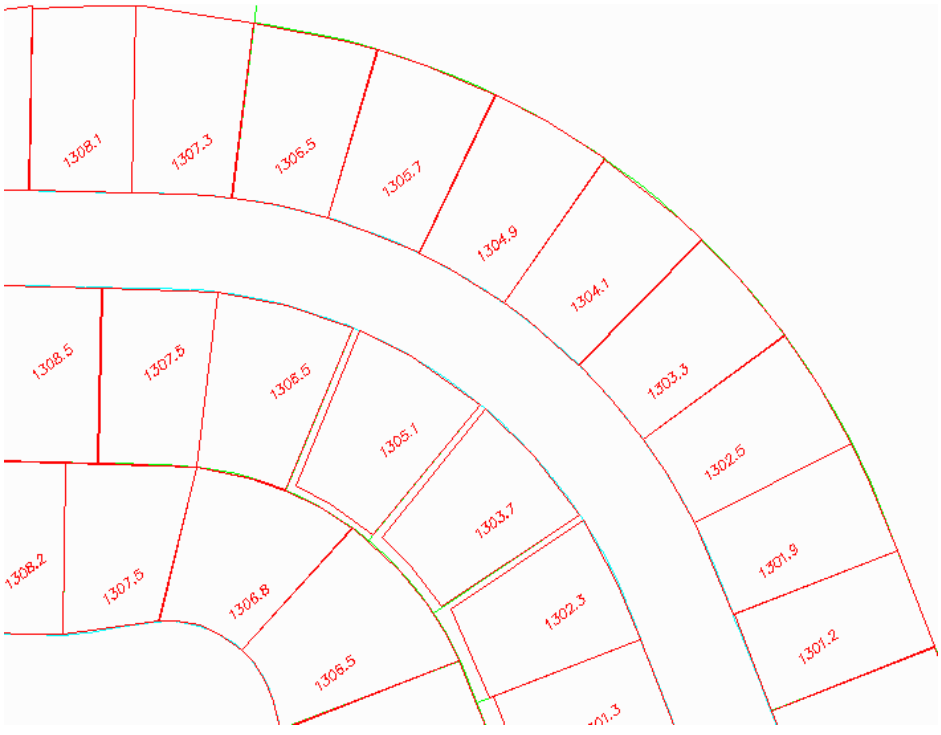


This dialog box allows you to create a new layer with the correct x,y coordinates and elevations. If the pad shares the same coordinates with other linework with different elevations than this dialog box allows you to offset the new polyline to avoid the problem of shared occupied points with different elevations. You can choose to have an interior offset or an exterior offset and also decide how much to offset the new polyline. Selecting Both will give both the interior pad elevation and the exterior contour elevations. This helps the transition from you pad elevation to the design contouring. The Snap Tolerance field joins linework which falls within the range you set to create a pad. Trim Outside Elevated Polylines will trim out contour elevations that go through your pad that you are not using elevations from within the pad.
Elevation to add to text values adds to the values from the elevation labels. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing.

After running the command you will be prompted to select the layers you want to use for the pad elevation and for the boundry of the pad. Sometimes pads are drawn with linework from two different layers and Carlson TakeOff allows you to pick all the correct linework. In addition, if your text has multiple Prefixes and Suffixes you will be prompted to select the ones you want to use the elevation from.



After clicking <OK> select all the pads and their elevation labels that you wish to change, press <Enter>, and the new layer with elevations will be created and placed in the Design target.



Prompts

Command: pad_by_text

Select layer sample of elevation text: Pick a label text
Selected text layer —-TX07
Select layer sample of boundary linework:
Selected linework layer PAD
Select another layer sample of boundary linework (Enter to continue):

Select text and linework to process.
Select objects: 1 found

Select objects: 1 found, 2 total

Select objects:

Analyzing entire selection...
Set elevation for 1 polylines.

Prerequisite: Pad polylines and elevations

Keyboard Command: pad_by_text

Drape 3D Polyline On Surface

Function

This command converts a 2D polyline into a 3D polyline by calculating 3D polyline vertices at all the intersects of the 2D polyline with surface entities (contour polylines, triangulation lines) and by interpolating elevations from these intersections at the original vertices locations. An application for this command is to create breaklines. For example, a ridge breakline could be generated from contour lines by drawing a 2D polyline along the ridge and across the contours. Then this command could grab the contour line elevations along the polyline to make a ridge breakline.

In addition to using entities in the drawing, the 2D polyline can be converted to 3D using a surface model stored in triangulation (.flt or .tin) file. If you use a file, then you can also use the polyline's current elevation as a vertical offset from surface.

Prompts

By Screen Entities:
Source of surface model [File/<Screen>]? *Type S for Screen*
Select polylines to convert.
Select objects: *select the polyline(s) to convert*
Select surface 3DFaces, lines and polylines.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Reading points ... 692

Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.
Converting polylines ...
Converted 1 polylines.

By a .flt or .tin File:
Source of surface model [<File>/Screen]? *Type F for File*
Select polylines to convert.
Select objects: *select the surface entities (contour polylines, breaklines, triangulation lines, etc)*

Use current polyline elevations as vertical offset from surface [Yes/<No>]? *Press Enter*
This will offset the new polyline by its current elevation. That is, if a polyline has an elevation of -4 and the surface you are converting it to has an elevation of 800, then saying Yes will drape the polyine at an elevation of 796.
Keep existing polylines [Yes/<No>]? *Press Enter*
This command creates a new 3D polyline, and this prompt allows you to keep the old polyline.
Set layer name for converted polylines [Yes/<No>]? *Press Enter*
This allows you to assign the new polyline to a layer.

Prerequisite: A polyline and surface lines or grid file or triangulation file.

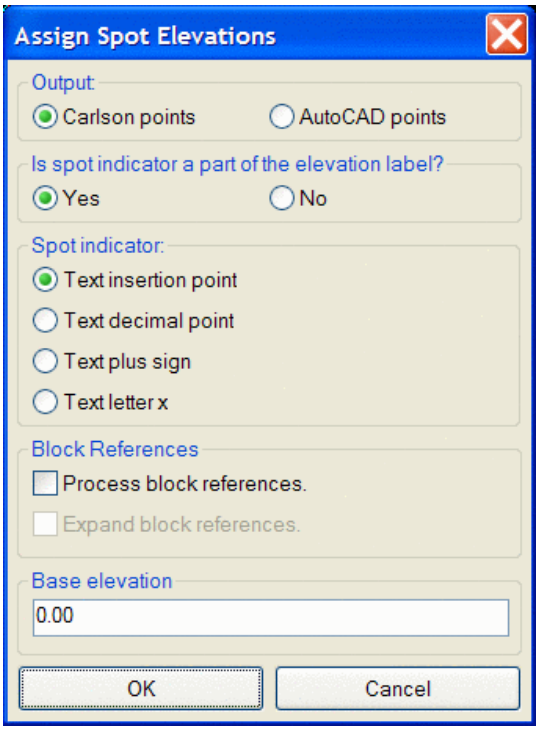
Keyboard Command: 2dto3dp

Convert Spot Elev To Points

Function

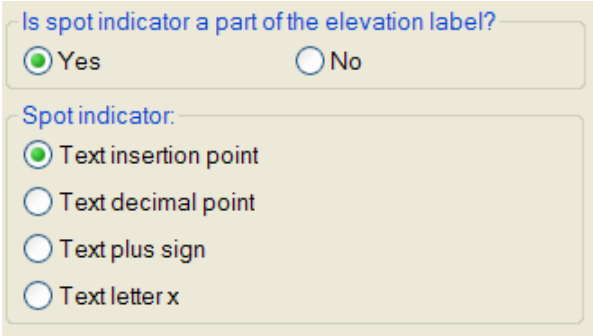
This command takes spot elevation entities with zero elevations and assigns them elevations according to corresponding elevation labels.

This dialog box allows you to choose the format of the spot elevations entities that you want to convert.



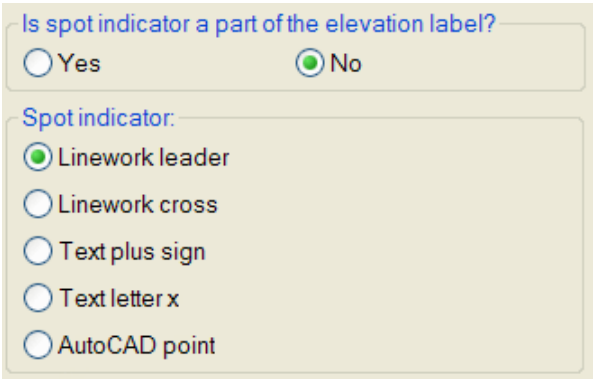
Output:
Carlson points: creates Carlson points at elevation of spot and stores them in coordinate file
AutoCAD points: creates AutoCAD point objects at elevation of spot

Is spot indicator a part of the elevation label?
If set to ''Yes'', four choices for Spot indicator are available to select from:



Text insertion point: uses the insertion point of the text for the location of the new point
Text decimal point: uses the decimal point in the text for the location of the new point
Text plus sign: uses the plus sign in the text for the location of the new point
Text letter x: uses the letter x in the text for the location of the new point

If set to ''No'', five choices for Spot indicator are available to select from:



Linework leader: creates a data point at the end of a leader

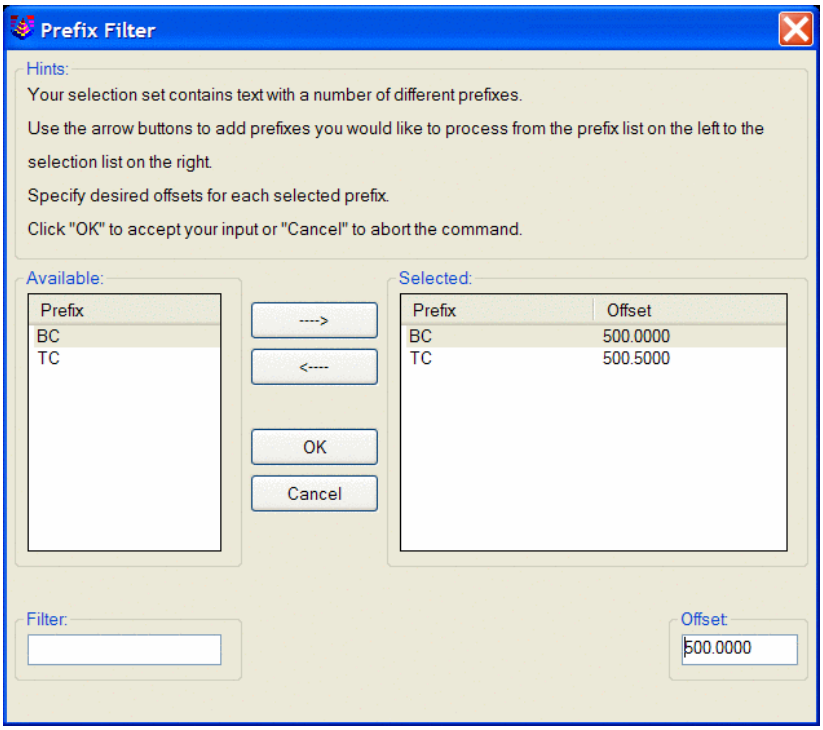


Linework cross: creates a data point at the intersection of a linework cross
Text plus sign: creates a data point at the insertion point of a text plus sign
Text letter x: creates a data point at the middle of a text letter x
AutoCAD point: creates a data point at the node of an AutoCAD point

Block References:
Process Block References: If check box is cleared, Carlson Civil searches only text entities for elevations, but if checked, Carlson Civil will search block references for elevations that are stored as attributes of a block. Use this option if the elevation is an attribute and the symbol designating the location of the spot elevation are both part of the block definition.
Expand Block References: Use this option to search block references when the elevation is stored as an attribute of a block, but the symbol designating the location of the spot elevation is a different block or even other geometry that is not defined within a block.

Base elevation: The value entered here is added to the existing spot elevations for all newly created points. Often times elevations are abbreviated to save time and space. If every elevation in a drawing is in the 500s instead of labeling every elevation 523.5, 543.3, 537.2 sometimes they are listed as simply 23.5, 43.3, 37.2. This command allows you to add a given amount, such as 500, to every label elevation to produce the correct elevation in the drawing. Note: The base elevation will not be added to any elevations that are closer to the base elevation value than they are to 0; e.g. if a base elevation of 500 is specified, 500 will be added to elevations like 23.4, 45.5, etc, but will not be added to elevations like 456.4 or 468.9.

Prefix Filter: Carlson Civil examines all selected spot elevations for prefixes or suffixes. If they are all the same, the command proceeds, but if there are different prefixes and/or suffixes found, the Prefix Filter dialog box is invoked. This dialog box allows you to select which prefixes and/or suffixes to use to create spot elevations, and also allows you to use different offset values for each. .



Prompts

<>**Starting point number <1>:** *press Enter*
Select TEXT, MTEXT spot elevations to process and any associated leader lines:
Select objects: *pick entities to process*
Pre-processing entity #40 of 40...
Filtering text entities
Processing elevation #40...
Converted 40 spot elevations.

Keyboard Command: spotelv2
Prerequisite: Spot elevations
</>

Assign Contour Elevations - Multiple in Series

Function

This command can be used to quickly and accurately assign the elevation of series ofAutoCAD polylines that have been converted from raster or digitized without correct elevations. Theroutine will automatically assign elevations to the polylines crossing the fence line selected by two points.At the same time the elevations are changed, the program can assign it a new layer, color,linetype, and polyline width. This process usually works best if contours are in a temporary (white)layer to start. When they are processed, they will take on the color of the new layers making it easyto distinguish which polylines have been processed.

Prompts

Settings/First Point: *(PressS to change settings or pick first point.)*

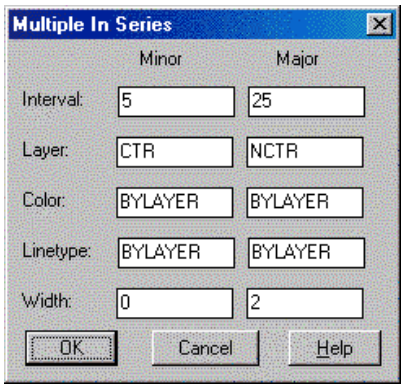
Second Point: *(Pick second point)*

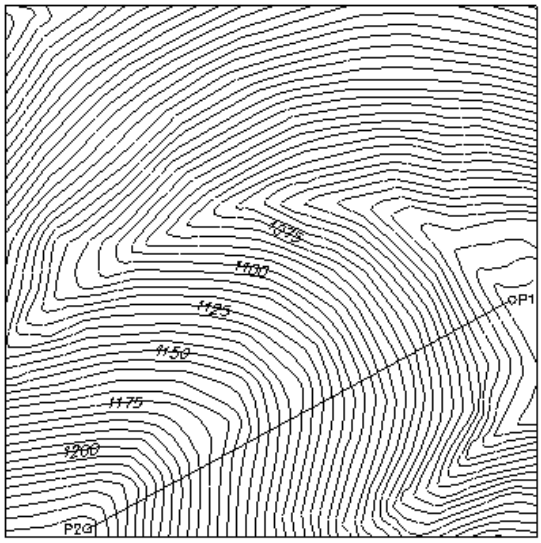
Beginning Elevation <0.00>: *1020*

Increment Direction U/D <U>: *(enter)*

Prerequisite: digitized polylines

Keyboard Command: grpcelev

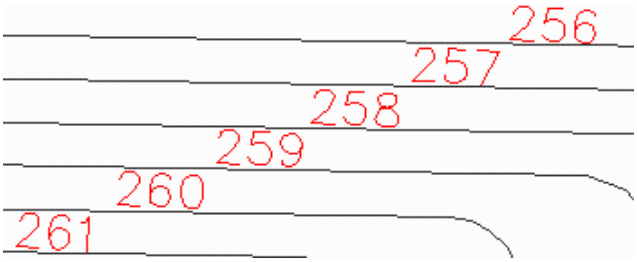




Assign Contour Elevations - From Contour Labels

Function

This command allows you to set elevations to contours from elevation labels.



Select a sample of the elevation text to be used on the contouring. Next, select a sample of the contouring that you want to add the elevations to. Now select all the contours and their corresponding elevation labels and press <Enter>. Carlson TakeOff will then add elevations to all the contours. You may be prompted to distinguish what contour goes with what elevation label. You can either press <Enter> to accept the contour that Carlson TakeOff has selected or you can Press <N> to choose another contour.

Prompts

Command:
TXTCELEV
Select sample of elevation text:
Select sample of a contour line:
Select contour lines and elevation text to process.
Select objects: all
5049 found
4041 were filtered out.

Select objects:

Joining adjacent polylines...

Reading the selection set ...
Joining ...
Pre-processing entity #1008 of 1008
Filtering text entities
Processing elevation text #518

Conflict detected: pick contour corresponding to current elevation text
Press N for next selection or Enter to accept current:
Remaking polyline #311

Prerequisite: contours and contours labels

Keyboard Command: TXTCELEV

Assign Contour Elevations - Single Elevation Group

Function

This command changes the elevations of polylines and can be used to set the elevations of contour polylines. The routine begins at a specified elevation and prompts for a selection set of polylines to set to the elevation. Then the routine repeats using the last elevation plus the elevation increment. Enter an empty selection set to exit the routine.

Prompts

Starting elevation <0.0>: *500.0*

Contour interval (negative for down) <1.0>: *5.0*

Select polylines to set to elevation 500.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 505.0.

Select objects: *pick the polylines*

Select polylines to set to elevation 510.0.

Select objects: *Press Enter*

Prerequisite: polylines

Keyboard Command: setcelev

Edit Contours

Function

This command revises a segment of a contour polyline. Begin by picking a point on the contour where you want to start editing. Then pick new points for the polyline. When finished picking new points, press Enter and then pick a point on the contour to connect with the new points. The polyline segment between the start and end points is then replaced with the new points.

Prompts

Select contour to edit: *pick the contour polyline at the place to start editing*

Pick intermediate point (Enter to End): *pick a point*

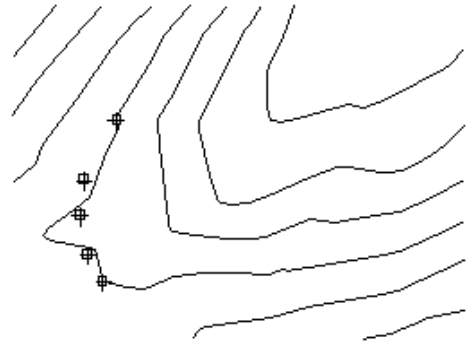
Pick intermediate point ('U' to Undo, Enter to End): *pick a point*

Pick intermediate point ('U' to Undo, Enter to End): *Press Enter*

Pick reconnection point on contour: *pick the contour polyline at the place to join*

Prerequisite: polylines with elevation (contour polylines)

Keyboard Command: editctr





Edit this contour by picking new points
Contour with segment replaced with new points

Inquiry Menu 10

List

Function

This command lists the object type, object layer, and*X*,*Y*,*Z* position relative to the current user coordinate system (UCS) and whether the object is in model space orpaper space.

The List command reports color, linetype, and lineweight information if these items are not set to BYLAYER. The thickness of an object is displayed if it is nonzero. *Z* coordinateinformation defines the elevation. If the extrusion direction of the entry differs from the *Z* axis (0,0,1) of the current UCS, the List command also reports the extrusion direction in UCS coordinates. The List reports additional information related to the specific object selected.

Prompts

Command:
LIST
Select objects: 3 found, 1 group

Select objects:

BLOCK REFERENCE Layer: "PNTS"
Space: Model space
Handle = 1F3D
Group = ∗A1
"SPT4"
at point, X=6135023.7190 Y=2190074.2098 Z= 800.0000
X scale factor 5.0000
Y scale factor 5.0000
rotation angle 0d0'0"
Z scale factor 5.0000

BLOCK REFERENCE Layer: "PNTS"
Space: Model space
Handle = 1F4D
Group = ∗A1
"SRVPNO1"
at point, X=6135023.7190 Y=2190074.2098 Z= 800.0000
X scale factor 5.0000
Y scale factor 5.0000
rotation angle 0d0'0"
Z scale factor 5.0000

ATTRIBUTE Layer: "PNTNO"
Space: Model space
Handle = 1F4E
Style = "PTXT"
Font file = TXT
center point, X=6135023.7190 Y=2190077.9598 Z= 800.0000
height 5.0000

value 1
tag PT#
rotation angle 0d0'0"
width scale factor 1.0000
obliquing angle 0d0'0"
flags normal
generation normal

ATTRIBUTE Layer: "PNTELEV"
Space: Model space
Handle = 1F4F
Style = "PTXT"
Font file = TXT
start point, X=6135031.2190 Y=2190071.7098 Z= 800.0000
height 5.0000
value 800

Prerequisite: an entity

Keyboard Command: LIST

Point ID

Function

This command allows you to display at the command line the coordinate values of a point you pick on the screen.

Prerequisite: None

Keyboard Command: PT_ID

Layer ID

Function

This command allows you to report at the command line the layer name of the selected entity.

Prompts

1 Pick entity to read layer: **pick an entity**

Layer: **FINAL**

2 Pick entity to read layer: **Press Enter**

Enter ends the command

Prerequisite: None

Keyboard Command: LAYERID

Curve Info

Function

This command displays information about a curve/arc. You can define the curve by selecting an arc entity or polyline arc segment, or you can pick three points on the arc, either by specifying point numbers or picking on the screen. The curve data is displayed in the text window. Press Enter to return to the graphics window.

Prompts

1 Define arc by, Points/<select arc or polyline>: **pick arc or polyline arc segment**

The curve information is displayed in the text window.

2 Press ENTER to continue. **Press Enter**

Prerequisite: None

Keyboard Command: CURVINFO

Polyline Info

Function

This command reports the length and elevation of the selected line or polyline.

Prompts

1 Pick Polyline or Line: **pick a polyline**

Polyline length: 145.43 Elevation: 100.0

Prerequisite: None

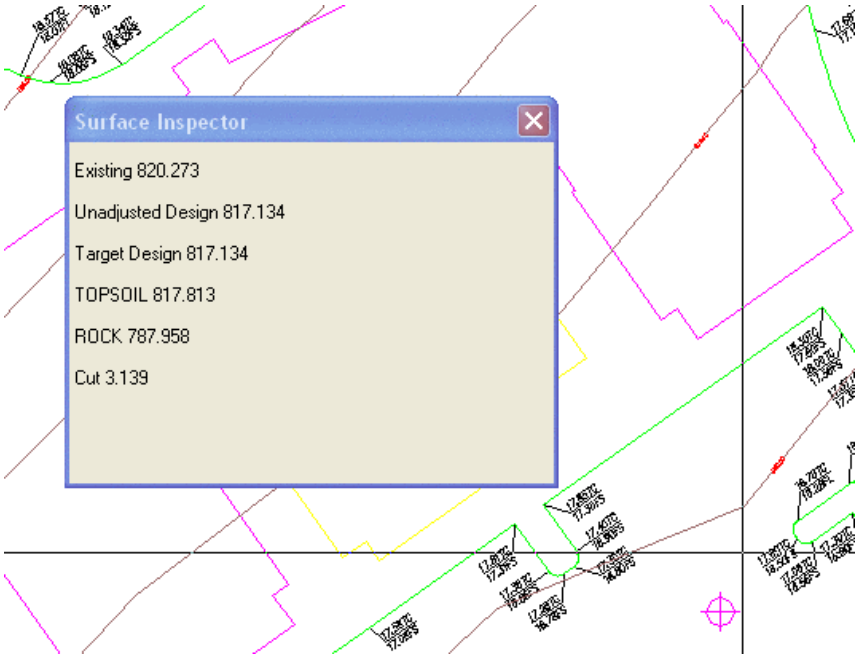
Keyboard Command: POLYLEN

Surface Inspector

Function

This command allows you to report and optionally label elevations from your drawing. You can analyze all of your different surface files at one time.

After running the command, Surface Inspector will begin showing you real-time elevations for each surface as you move the cursor on the screen. If you pick apoint or enter coordinates, the elevation will be labeled along with the surface name.



Surface inspector shows you real-time elevations as you move the cursor over your surface.

Prerequisite: Surface Model(s)

Keyboard Command: surfvals

Surface Report

Function

This command reports a variety of information on each of your different surfaces. This is useful for checking for bad data and the file names of your surfaces. An example is below.

Surface Report 3/10/2005 15:34
Max Cut: 18.327 at 409269.984,207196.674
Max Fill: 1.943 at 409389.586,207248.866

Original Ground After Topsoil Removal
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-ex.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

Design With Subgrade and Topsoil Replacement
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-fn.flt
Date Modified: Thu Feb 10 10:02:08 2005

File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

Original Ground Before Topsoil Removal
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-og.flt
Date Modified: Thu Feb 10 10:02:05 2005
File Size: 64,028
Points: 259, Edges: 744, Triangles: 486
Min Z: 184.000 at 409299.790,206879.287
Max Z: 210.000 at 409571.562,207177.240

Design Without Subgrade or Topsoil Replacement
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-bs.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

Design With Subgrade
File: C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo3-zn.flt
Date Modified: Thu Feb 10 10:02:08 2005
File Size: 153,038
Points: 609, Edges: 1,779, Triangles: 1,171
Min Z: 176.000 at 409357.096,206821.604
Max Z: 206.000 at 409551.532,207185.124

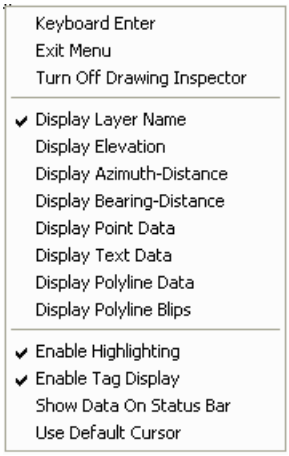
Prerequisites: A Surface

Keyboard Command: SURF_STATS

Drawing Inspector

Function

This command controls the reporting of object properties when you move the cursor over an entity. You activate the Drawing Inspector by selecting the Drawing Inspector command from the Inquiry menu, then you right click to display the menu containing a list of properties you can display and a list of display options. The available properties are: Layer Name, Elevation, Bearing-Distance, Point Data, Text Data, Polyline Data, and Polyline Blips. After you select a property, you move the pointer over an entity, and the selected property is displayed either in a pop-up window next to the pointer and/or on the status bar, depending on the selected display option.



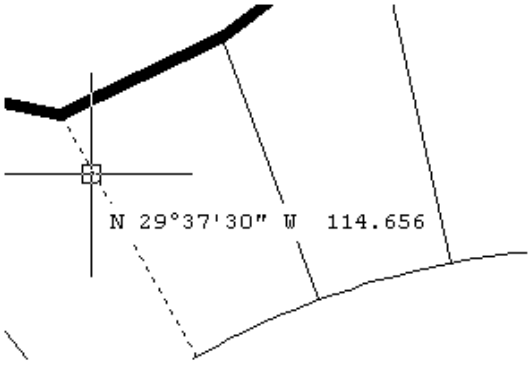
1 In the Drawing Inspector menu, you can choose one or more properties to display.

- **Display Layer Name**: Allows you to display the layer name of the entity.
- **Display Elevation**: Allows you to display the elevation of the entity.
- **Display Bearing-Distance**: Allows you to display the bearing and distance of a line.
- **Display Point Data**: Allows you to display the coordinate data of point.
- **Display Text Data**: Allows you to display the attributes of text.
- **Display Polyline Data**: Allows you to display the end point elevations, horizontal distance, slope distance and slope ratios.
- **Display Polyline Blips**: Allows you to display the vertices on polylines.

2 In the Drawing Inspector menu, you can choose how the property information is reported.

- **Enable Highlighting**: Allows you to highlight the object that the Drawing Inspector is reporting.
- **Enable Tag Display**: Enables you to view the information next to the cursor on the screen.
- **Show Data On status Bar**: Enables you to view the information on the status bar, in the lower right corner of the screen.
- **Use Default Cursor**: When enabled, you see only the CAD cursor. When disabled, mouse pointer is also shown.

Other commands can be run as usual while Drawing Inspector is active. To turn off the Drawing Inspector, pick the Drawing Inspector command from the menu a second time.



Prerequisite: None.

Keyboard Command: INSPECTOR

Layer Inspector

Function

This command allows you to edit the properties of different layers.



All the layers in your drawing are listed in the left-hand side of the command panel. When you select a layer Entity Count reports the number of entities in the layer. This command allows you to erase those entities, rename the layer, purge the layer, or make the layer current. Purging the layer will remove it from the drawing. All entities must be erased before you purge a layer.
Toggle on Zoom, Isolate, and/or Highlight to do just that when you select a layer. Use the magnifying glass and the arrows to change the view of yourdrawing. Restore View On Exit will return you to the view you where in when you began the command.

Prerequisite: none

Keyboard Command: layer_inspect

Layer Report

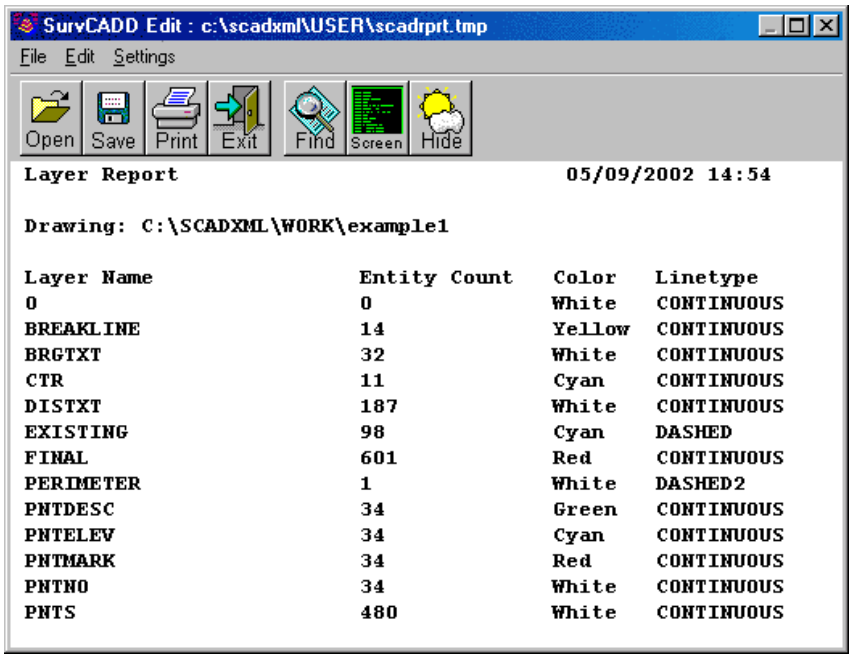
Function

This command generates a report containing each layer name, the number of entities on each layer, and the color and linetype applied to each layer.

Pull-Down Menu Location: Inquiry

Prerequisite: None.

Keyboard Command: layerid


SurvCADD Edit : c:\scadxml\USER\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Layer Report 05/09/2002 14:54
Drawing: C:\SCADXML\WORK\example1
Layer Name Entity Count Color Linetype
0 0 White CONTINUOUS
BREAKLINE 14 Yellow CONTINUOUS
BRGTXT 32 White CONTINUOUS
CTR 11 Cyan CONTINUOUS
DISTXT 187 White CONTINUOUS
EXISTING 98 Cyan DASHED
FINAL 601 Red CONTINUOUS
PERIMETER 1 White DASHED2
PNTDESC 34 Green CONTINUOUS
PNTELEV 34 Cyan CONTINUOUS
PNTMARK 34 Red CONTINUOUS
PNTNO 34 White CONTINUOUS
PNTS 480 White CONTINUOUS

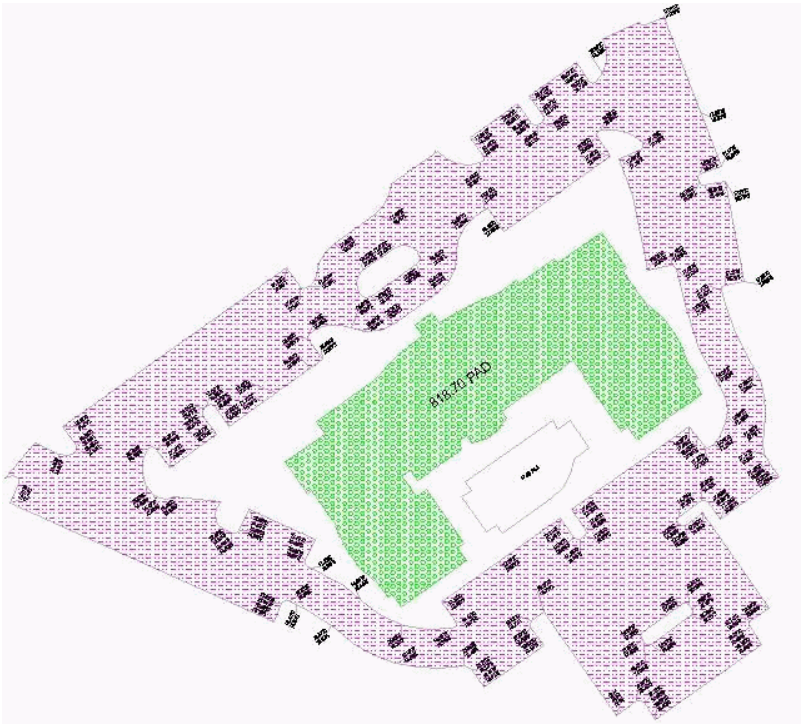
Hatch Subgrade Areas

Function

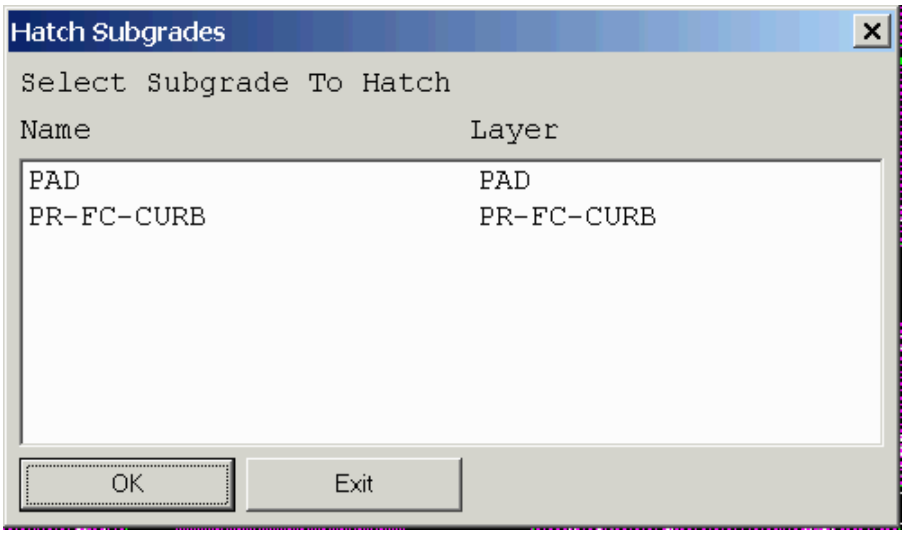
This command draws a hatch with a specified color and pattern for the area that the selected subgrade area applies to. The purpose is to allow you to visually review a subgrade area to make sure that the area coverage is correct.

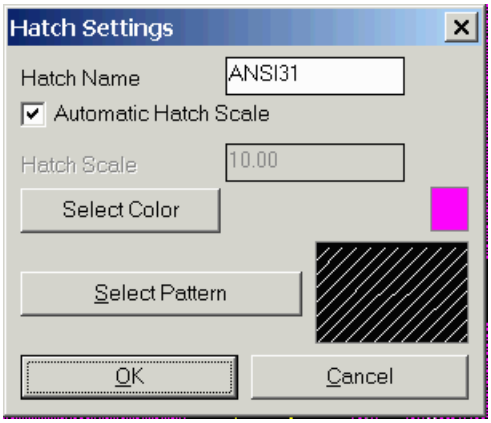
The command displays a dialog to select which subgrade to hatch. The list of available subgrades comes from the layers with subgrade depths as set in the Define Layer Target/Material/Subgrade command. Then there is a dialog for the hatch pattern, color and scale. The scale determines how spread out the pattern is within the hatch. The Automatic Hatch Scale option checks the size of the subgrade areas and sets the scale to make the pattern fit.

The resulting hatch areas show where the subgrade is applied. In the example below, notice how the islands are not hatched because they are curb polylines that are already inside another curb polyline. Also note that the smaller pad area is not hatched because this polyline layer is different than the bigger pad polyline.


818.70 PAD

Prompts


Hatch Subgrades
Select Subgrade To Hatch
Name
Layer
PAD
PAD
PR-FC-CURB
PR-FC-CURB
OK
Exit



Prerequisite: subgrades

Keyboard Command: hatch_subgrade

Erase Subgrade Hatches

Function

This command removes from the screen the subgrade hatches created by the commandHatch Subgrade Area.

Prompts

Command: erase_subgrade

Prerequisite: hatch subgrade areas

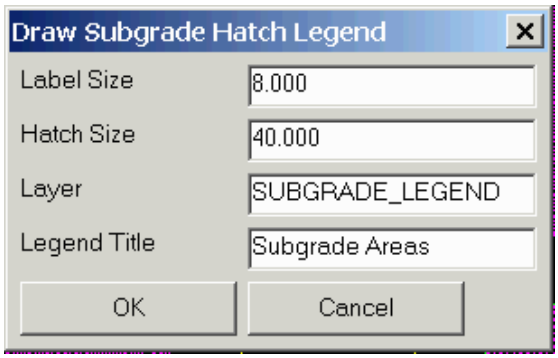
Keyboard Command: erase_subgrade

Hatch_Subgrade_Areas

Function

This command draws a legend for the subgrade areas currently in the drawing. The legend includes the subgrade names and squares of the hatch patterns. The size of the labels, size of the hatch squares, layer for the legend entities and the legend title are set in the dialog shown below. The subgrade hatches to include in the legend are automatically selected from all the subgrade hatches currently in the drawing that were created by the Hatch Subgrade Areas command.

Prompts



Prerequisite: hatched subgrade areas

Keyboard Command: draw_subgrade_legend

Label Subgrade Areas

Function

This command lets you label the subgrade depth and area (in sq. ft. or meters). The label is placed at the center of the subgrade area, but can be moved with the Move command under Edit.

Pull-Down Menu Location: Inquiry-> Subgrade Areas

Prerequisite: Subgrade Areas

Keyboard Command: label_subgrade

Erase Subgrade Labels

Function

This command erases subgrade labels.

Pull-Down Menu Location: Inquiry-> Subgrade Areas

Prerequisite: subgrade labels

Keyboard Command: erase_subgrade_labels
nil

List Elevation

Function

This command displays the elevation of a polyline or line. With a 3D polyline, the elevation ofthe 3D polyline at the pick point is reported along with the elevation of each vertex. See also,the *Drawing Inspector* command on the ∗Inq-Set∗ menu.

Prerequisite: an entity

Keyboard Command: LSTELEV

Distance Report

Function

This command reports the horizontial distance, slope distance, and elevation difference between two points.

Prompts

Pick point or enter point number:
(409375.0 207039.0 0.0)
Pick second point or enter point number:
(409400.0 207082.0 0.0)
Horiz Dist: 49.73 Slope Dist: 49.73 Elv Diff: 0.00

Prerequisite: none

Keyboard Command: distrprt

Bearing and Distance

Function

This command reports the slope distance, slope ratio, bearing, azimuth and vertical angle between two 3Dpoints. Pick or enter the coordinates of two points or select a line or polyline segment to calculate between the segment endpoints.

Prompts

1 Specify bearing-distance from(Line/PLine/<Points>)? **Press Enter**

2 Pick point or enter pointnumber: **pick a point**

3 Pick second point or enterpoint number: **pick a point**

The horizontal distance, slope distance, elevation difference, vertical angle, slope, bearing, and azimuth are reported.

Prerequisite:None

KeyboardCommand: 3DIST

Label Angle

Function

This command will label and report the interior and exterior angles between two directions. The angles can be defined by three points or by two line or polyline segments that have a common endpoint.

Prompts

Define angle by, Points/<select line or polyline>: *pick a polyline segment*

Select adjoining line or polyline: *pick another polyline segment*

Interior: 72d39'46'' Exterior: 287d20'14''

Angle to label (<Interior>/Exterior/None)? *press Enter*

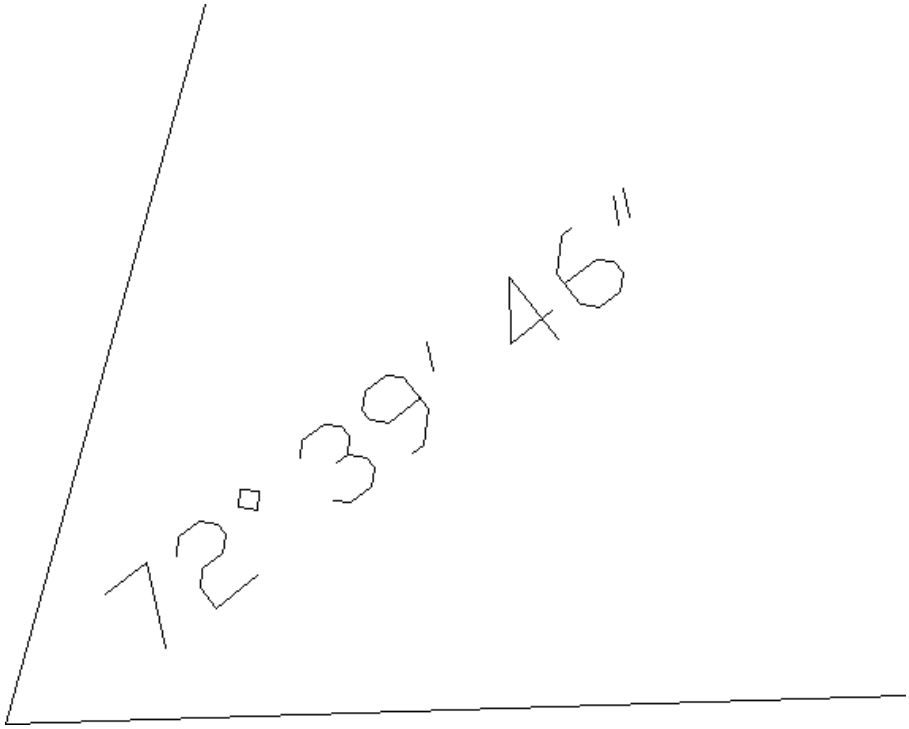
Define angle by, Points/<select line or polyline>: *press Enter to end*

Pull-Down Menu Location: Annotate

Prerequisite: None

Keyboard Command: labang

File Name: \lsp\angle.lsp



Polyline Report

Function

This command generates a report of bearing-distance and curve data for all the point along the selected polyline. The closure is reported between the starting and ending points of the polyline. The polyline area is also reported. After executing the command, by pressing O for options, report deflection angles can be turned on by selecting Y for yes at the prompt. The deflection angle will be displayed in the report along with the bearing and distance of the polyline segments.

Prompts

Starting station <0.0>: *Press Enter*

Options/Select polyline to report: *pick a polyline*

Standard Report Viewer Displays the report for the selected polyline.

Options/Select polyline to report (Enter to End): *Press Enter*

Keyboard Command: plreport

Pull-Down Menu Location: Tools, Polyline Tools

<>**Prerequisite:** a polyline

Polyline Report 05/22/2002 13:24 </>

NORTHING EASTING STATION BEARING DISTANCE

5790.690 4088.812 0.000

N 49°52'11'' E 455.737

6084.426 4437.259 455.737

RADIUS: 367.183 LENGTH: 725.711 CHORD: 613.228 DELTA: 113°14'28''

CHORD BRG: S 73°30'35'' E PC-R: S 40°07'49'' E PT-R: S 73°06'39'' W

RADIUS POINT: 5803.685,4673.919

5910.360 5025.265 1181.448

S 16°53'21'' E 851.945

5095.161 5272.774 2033.393

S 52°00'05'' W 548.996

4757.175 4840.152 2582.389

N 54°19'05'' W 857.964

5257.614 4143.257 3440.353

RADIUS: 695.425 LENGTH: 430.350 CHORD: 423.516 DELTA: 35°27'23''

CHORD BRG: N 72°02'46'' W PC-R: S 35°40'55'' W PT-R: S 00°13'33'' W

RADIUS POINT: 4692.743,3737.624

5388.163 3740.364 3870.702

N 40°52'52'' E 532.394

5790.690 4088.812 4403.097

Closure Error Distance> 0.000

Total Distance> 4403.097

Polyline Area: 1201606.5 sq ft, 27.59 acres

Slope Report

Function

This command calculates the sloped surface area, average slope and average elevation on a site. The surface is defined by a (.tin) file or (.flt) file.

For the either option, there are options to specify inclusion and exclusion perimeters. When inclusion perimeters are specified, only the area within the inclusion perimeters is calculated.The area within exclusion perimeters is not included in the calculations. Inclusion and exclusion perimeters are represented by closed polylines and must be drawn prior to calling this routine.

Prompts

Command:
SLOPE_REPORT
Process original ground or design surface [Original/<Design>]? *D* for Design and then pick the surface file for the site.

Loading edges...
Loaded 1726 points and 4836 edges
Created 3111 triangles

Select the Inclusion perimeter polylines or ENTER for none.
Select objects: *1 found*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none.
Select objects: *Press Enter*

Average Slope Report

Slope Report 4/27/2006 14:59

Processing triangulation file: C:\Documents and Settings\Todd Carlson\Desktop\Testing\demo1-bs.flt

Horizontal Area: 22327.0 sq ft, 0.513 acres
Slope Area: 22327.8 sq ft, 0.513 acres
Average Elevation: 817.94
Average Slope: 0.66%

Prerequisite: A (.flt) file or (.tin) file

Keyboard Command: SLOPE_REPORT

Slope Zone Analysis

Function

This command calculates the surface area of a site in different slope zone ranges. Theprogram requires 3D Face entities that can be generated by the*Plot 3D Grid File* command or the Draw Triangulation Faces option of*Triangulate & Contour*. For each slope zone, the 3D Faces canbe hatched with any AutoCAD hatch pattern, including the SOLID pattern, or left empty withthe NONE pattern.

This command can also generate contours of the slope zones based on the calculated slope ateach point of the 3D Faces. The slopes can vary greatly between neighboring points. Whencontoured directly, these slope data points produce incoherent contours. Instead this routine appliesa filtering algorithm that reduces the noise. There is another option to output a grid file of theslope values.

There are also options to specify inclusion and exclusion areas. When inclusion areas arespecified, only the slope area within the inclusion polyline is calculated. Slope area within anexclusion polyline are not included in the calculations. Inclusion and exclusion areas are representedby closed polylines and must be drawn prior to calling this routine. Without inclusion andexclusion polylines, all the slope area of each selected 3D Face is used.

Prompts

Slope Zone dialog Specify the slope zones, colors and patterns from lowest to highest.

Draw contours of slope zones (Yes/<No>)? Yes

Output a grid file of the slope (Yes/<No>)? No

Apply hatch patterns to grid cells (Yes/<No>)? No. This option fills the 3D faces with the hatch pattern defined for the different zones. Without the hatch patterns the program will just modifythe 3D faces color.

Freeze grid layer after processing (Yes/<No)? No. This freezes the layer of the 3D faces so that the slope contours are easier to see.

Select Rectangular 3D Faces to Analyze...

Select objects: *C*

First corner: *(pick point)* **Other corner:***(pick point)* **1200 found**

Select objects: *[Enter]*

Select the Inclusion perimeter polylines or ENTER for none:
Select objects: *Pick a closed polyline for the limits of disturbed area*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Analyzing Entity> 1000 of 1000

Select point for color legend: *(pick point)*

If a point is picked a legend showing the color of each range is drawn. The legend is drawnaligned to the current view *UCS*. For this reason it is best to have the mesh at the*Vpoint* at which it will be plotted before executing the analysis program.

Contouring Options Dialog This dialog appears if the Draw Slope Contours option is active.You may want to change the Contour Layer so it doesn't conflict with any existing contour layers.The Hatch Slope Zones applies the hatch patterns to the slope zones between the slope contours.

Reading points 1000 ...

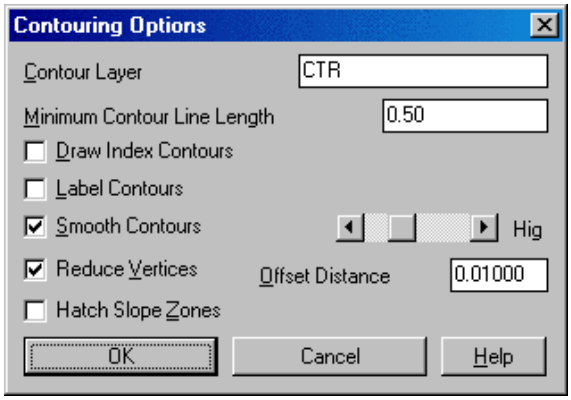
Processing points 1000 ...

Triangulating points 1000 ...

Inserted 4205 contour vertices.

Prerequisite: 3D Face entities plotted usually created by the*Make 3D Grid File* command.

Keyboard Command: szone



Slope Report

Number of 3DFaces analyzed> 1334

Surface % of

Zone Range Area S.F. Acres Total

———————

1 <= 10.00 19902.43 0.46 16.37

2 > 10.00 38169.69 0.88 31.39

3 > 20.00 35833.72 0.82 29.47

4 > 30.00 19622.12 0.45 16.14

5 > 40.00 8082.44 0.19 6.65

———————

Total Surface Area> 121610.41

Average Slope > 21.87

Minimum Slope%> 0.00 Maximum Slope%> 66.77

Slope Zone Analysis report



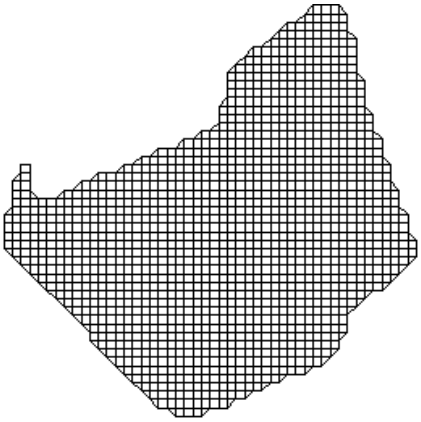


Figure 2
Figure 1

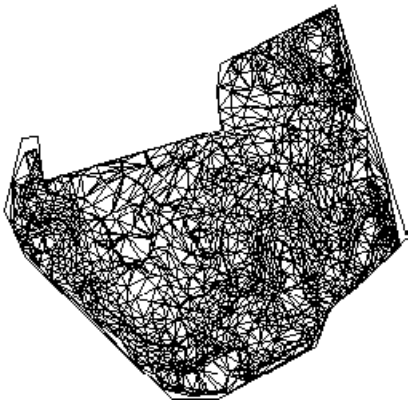
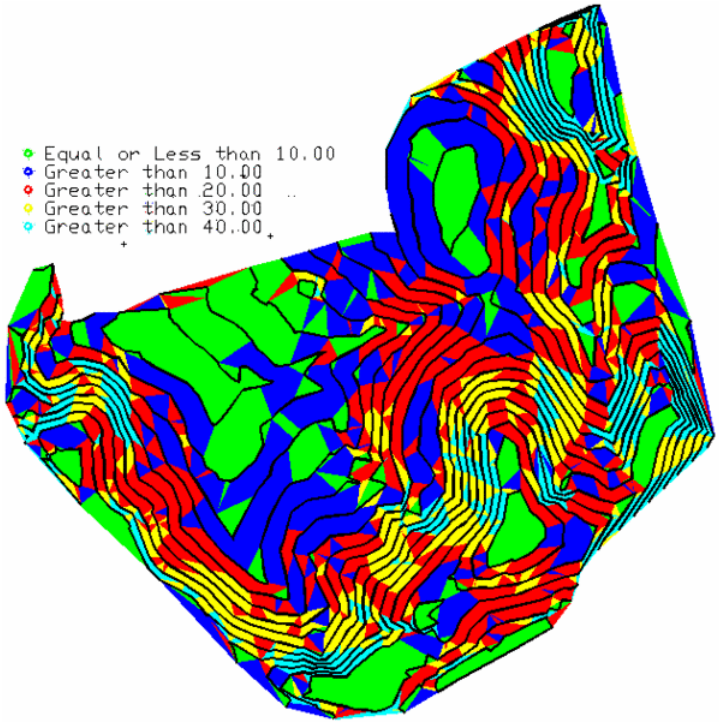




Figure 4
Figure 3



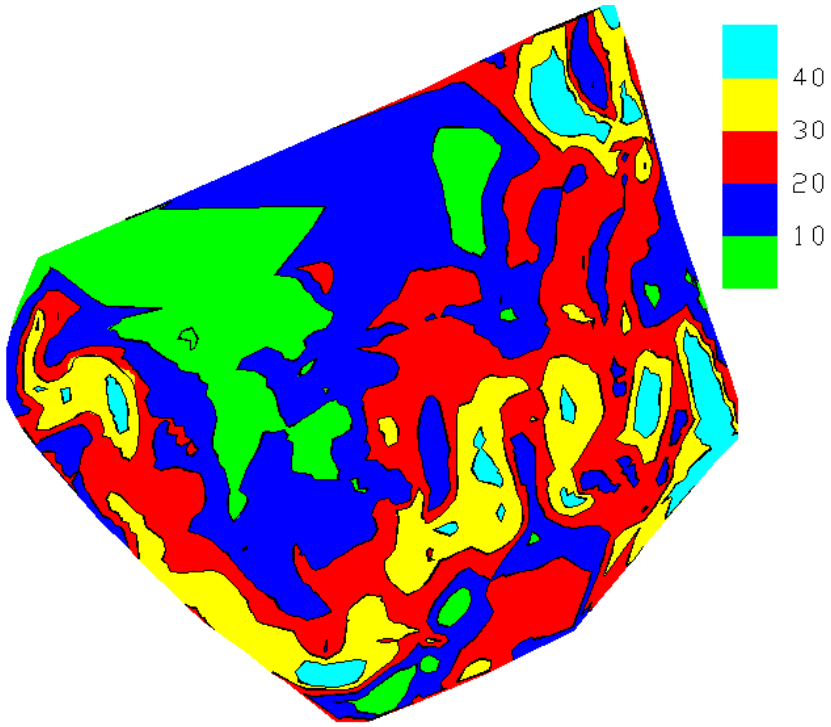


Figure 6
Figure 5
Figure 1: Surface contours

Figure 2: 3D Faces from a grid surface model created by*Make 3D Grid File* and drawn by *Plot 3DGrid File*.

Figure 3: 3D Faces created by *Triangulate &Contour* with the Draw Triangutation Faces option.

Figure 4: Slope zone contours created from the grid 3D Faces.

Figure 5: Slope zones that follow the surface contours using the triangulation 3D Faces.

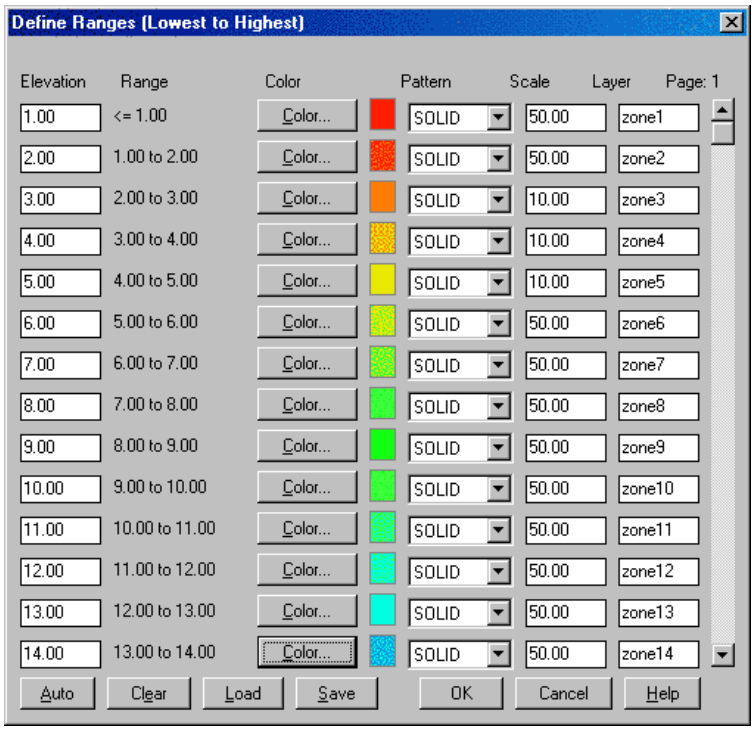
Figure 6: Hatched slope zone contours created from the grid 3D Faces.

Elevation Zone Analysis

Function

This command can be used to calculate the surface area of a surface in different elevation zone ranges. The program requires 3D Face entities. For each elevation zone, the 3D Faces can be hatched with an AutoCAD hatching pattern, solid filled with the SOLID pattern, or left empty with the NONE pattern. The 3D Faces are also placed in a layer for each zone.

There are also options to specify inclusion and exclusion areas. When inclusion areas are specified, only the area within the inclusion polyline is calculated. Areas within an exclusion polyline are not included in the calculations. Inclusion and exclusion areas are represented by closed polylines and must be drawn prior to calling this routine. Without inclusion and exclusion polylines, all the area of each selected 3D Face is used.


Define Ranges (Lowest to Highest)
Elevation
Range
Color
Pattern
Scale
Layer
Page: 1
1.00 <= 1.00 Color... SOLID 50.00 zone1
2.00 1.00 to 2.00 Color... SOLID 50.00 zone2
3.00 2.00 to 3.00 Color... SOLID 10.00 zone3
4.00 3.00 to 4.00 Color... SOLID 10.00 zone4
5.00 4.00 to 5.00 Color... SOLID 10.00 zone5
6.00 5.00 to 6.00 Color... SOLID 50.00 zone6
7.00 6.00 to 7.00 Color... SOLID 50.00 zone7
8.00 7.00 to 8.00 Color... SOLID 50.00 zone8
9.00 8.00 to 9.00 Color... SOLID 50.00 zone9
10.00 9.00 to 10.00 Color... SOLID 50.00 zone10
11.00 10.00 to 11.00 Color... SOLID 50.00 zone11
12.00 11.00 to 12.00 Color... SOLID 50.00 zone12
13.00 12.00 to 13.00 Color... SOLID 50.00 zone13
14.00 13.00 to 14.00 Color... SOLID 50.00 zone14
Auto
Clear
Load
Save
OK
Cancel
Help

Prompts

Elevation Zone dialog Specify the elevation ranges, colors and patterns.

Plot average elevation text in each grid cell (Yes/<No>): *Y*

Command: elvzone

Name of property represeted by Z value <Elevation>: *Press Enter*

Plot average Elevation text in each grid cell [Yes/<No>]? *Press Enter*

Subdivide grid cells at zone boundaries [<Yes>/No]? *Press Enter*

Select 3D Faces to Analyze...
Select objects:

First corner: *(pick point)* **Other corner:** *(pick point)* **1200 found**

Select objects: *[Enter]*

Select the Inclusion perimeter polylines or ENTER for none:

Select objects: *Pick a closed polyline for the limits of disturbed area*

Select objects: *Press Enter*

Select the Exclusion perimeter polylines or ENTER for none:

Select objects: *Press Enter*

Surface %of

Zone Range Area Total

_____

1 <= 100.00 26822.41 27.25

2 > 100.00 27439.15 27.88

3 > 110.00 16727.98 17.00

4 > 120.00 12357.28 12.55

5 > 130.00 11700.36 11.89

6 > 140.00 3380.98 3.43

_____

Total Surface Area> 98428.16

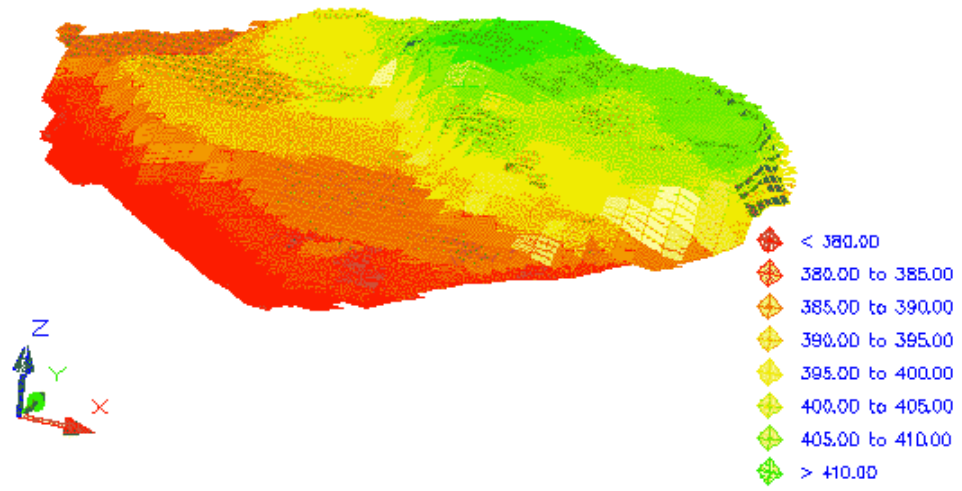
Average Elevation > 110.83

Minimum Slope% > 0.00 Maximum Slope%> 252.97

Minimum Elevation > 90.00 Maximum Elevation> 149.53

Select point for color legend: *(pick point)*

If a point is picked, a legend showing the color of each range is drawn. The legend is drawn aligned to the current view *UCS*. For this reason it is best to have the mesh at the *Vpoint* at which it will be plotted before executing the analysis program.



Result of Elevation Zone Analysis viewed in 3D and shaded.

Prerequisite: 3D Face entities plotted usually created by the *Make 3D Grid File* command.

Keyboard Command: elvzone

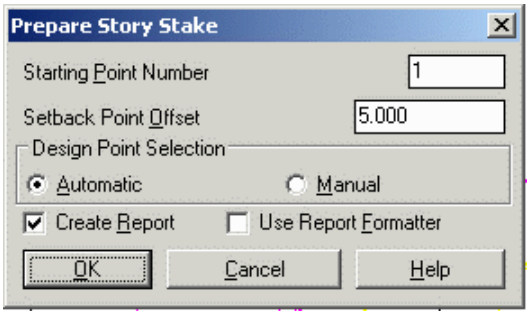
Story Stake From Surface Entities

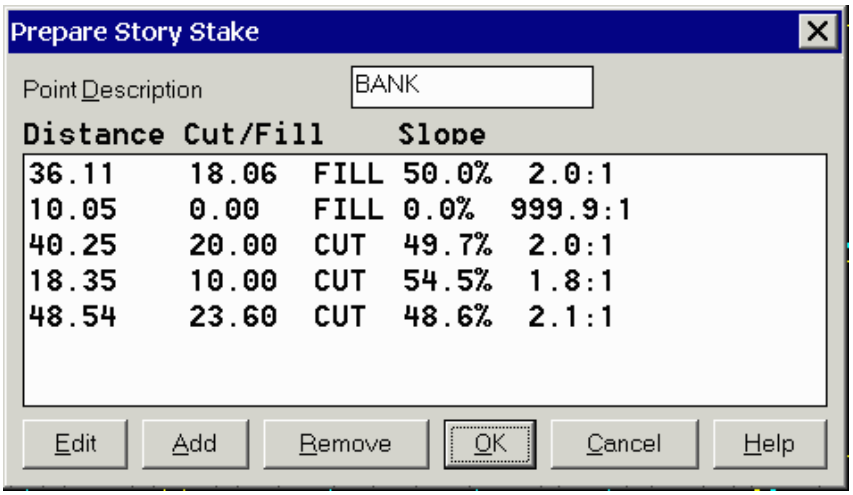
Function

This command creates points with cut/fill information stored in the note fields for the points. Beginning at a point and facing a specified direction, the cut/fill information describes a design surface that is defined by contours and 3D polylines in the drawing.

First, there is a dialog to set the starting point number, setback offset, selection mode and report options. The Automatic selection mode will create a story point for each crossing linework. The Manual mode will show the crossings and let you pick which ones to report.

The program prompts you to pick the starting point followed by a direction point. Then the intersections for all the contours and 3D polylines between these two pointsare calculated and the resulting horizontal distances and slopes are shown in a dialog. In this dialog, you can edit, add or remove these slopes descriptions. The Point Description can also be specified. When OK is clicked, a point in the coordinate file is created at the starting point with this information stored in the note file. An offset point is also created at the specified offset distance back from the starting point. At the end of Story Stake, a report of all the created points and the corresponding cut/fill data is shown if the Create Report option was set. Story Stake does not draw the points in the drawing. These points can be drawn using the Draw-Locate Points command.







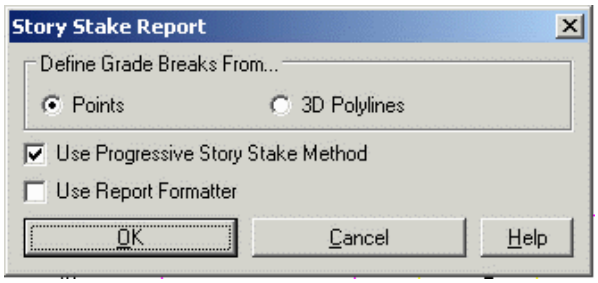
Story Stake By Points/Polyline

Function

This command creates a story stake report of distances, slope and cut/fill between selected points or along a selected 3D polyline.

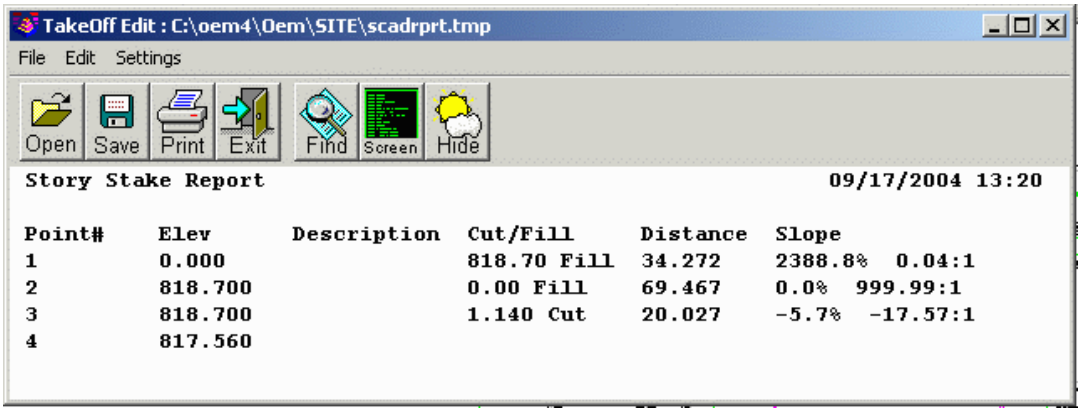
First, there is an options dialog. The Progressive method reports each distance from the previous point to the next. The non-progressive method reports the distances from the starting point to each point. The report formatter option

lets you customize the fields for the report and output the report to Excel.



For the points method, there are prompts at the command line to pick the points. The points should be picked one at a time in series from start to end. The points can be entered by point number from the current coordinate file or by screen pick.

For the polyline method, there is a prompt to select the 3D polyline for the grade. The polyline should be drawn in the order of the story stake.


TakeOff Edit : C:\oem4\Oem\SITE\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Story Stake Report 09/17/2004 13:20
Point# Elev Description Cut/Fill Distance Slope
1 0.000 818.70 Fill 34.272 2388.8% 0.04:1
2 818.700 0.00 Fill 69.467 0.0% 999.99:1
3 818.700 1.140 Cut 20.027 -5.7% -17.57:1
4 817.560

Area Defaults

Function

This command allows you to specify default settings for area labeling.

Sequence: The top portion of the dialog lists 10 area values. For each value, you can specify a value under the Order# column and a Suffix. The area label will include the values in the order specified. If an Order# is left blank, that value is not labeled.

Precision for Square Units Labels: Choose precision level for labeling Square Feet, Square Yards, Square Miles, Square Meters, and Square Kilometers.

Precision for Other Area Labels: Choose precision level for labeling Acres, Cuerdas, and Hectares.

Precision for Inverse with Area: Choose precision level when using Inverse with Area.

Label Both Feet & Meters for Inverse with Area: When this option is turned ON, both feet and meters will be shown in the *Inverse with Area* report.

Label Area with +/-: This allows you to display + or ± in the area labels.

Use Commas in Labels: This allows you to use commas in the area labels.

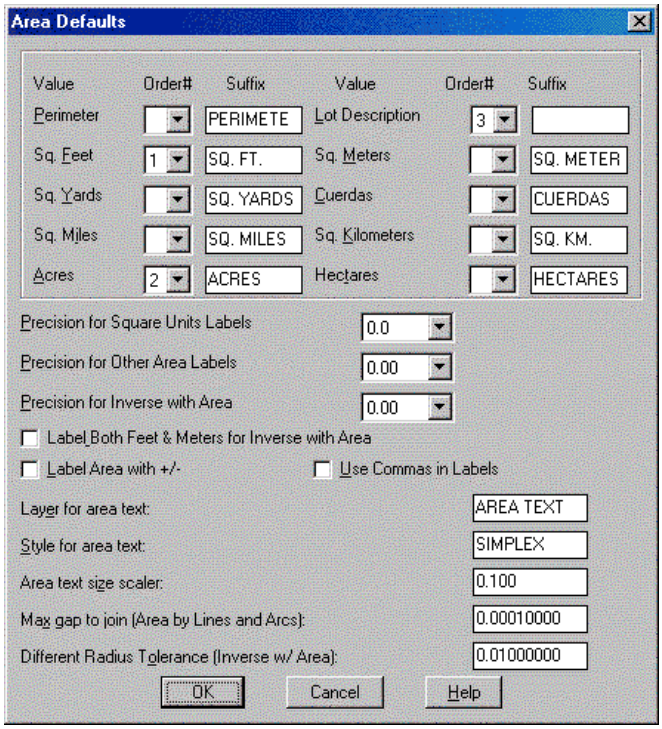
Layer for area text: This allows you to assign a layer for the area text.

Style for area text: This allows you to set a text style for the area labels.

Area text size scaler: This value is multiplied by the horizontal scale to obtain the actual text size.

Max gap to join (Area by Lines and Arcs): You use this option during *Area by Lines & Arcs* command. When connecting lines and arcs that define the perimeter, the program will join endpoints if the distance between the two points is less than the specified gap. Otherwise the program will report an error and will not report an area.

Different Radius Tolerance: You use this option to check the difference between the PC-Radius and PT-Radius on curves. If the difference between these distances is greater than this tolerance, an accurate area calculation cannot occur and the command displays a warning.



Prerequisite: None

Keyboard Command: defarea

Area by Inverse

Function

This command calculates and labels the area of the inverse of a drawn polyline.

Prompts

Command: IA

USER path is> C:/Program Files/Carlson TakeOff 2004// SN matches
Station/<Pick Starting point or point number>:
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end):
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): c
SQ. FEET: 4543.3 SQ. YARDS: 504.8 SQ. MILES: 0.0
ACRES: 0.10 PERIMETER: 308.87
Pick area label centering point:
Erase Polyline (Yes/No) <Yes>: Y

Prerequisite: Set Area Label Defaults.

Keyboard Command: IA

Area by Lines & Arcs

Function

This command allows you to calculate the area of a perimeter or lot defined by lines, arcs, or polylines. Default settings for this command are set in *Area Defaults*. One of these settings is Max gap to join. If there is a gap greater than this value, the area is not reported and the program will show where the gap is with a temporary X symbol.

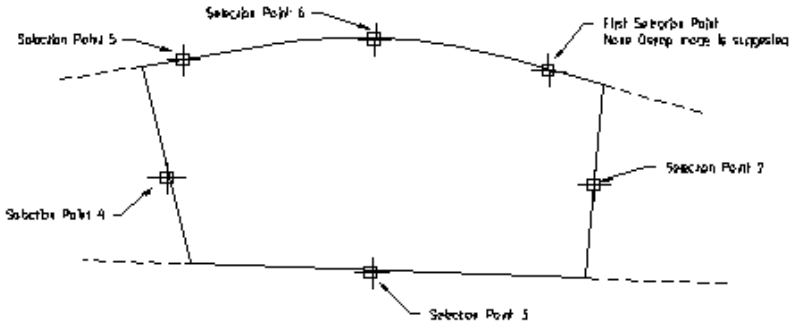
Prompts

Select lines & arcs which make up figure for area calculation.

Select Objects: *(select lines & arcs orpolyline)*

Lines & arcs are then joined together and the area calculated.

Enter/pick Label center point: *(pick point)*The area is then plotted at the point selected.



Prerequisite: lines, arcs, or polylines to define the area.

Keyboard Command: JOINAREA

Area by Interior Point

Function

This command calculates and labels the area of the perimeter surrounding a picked interiorpoint. The AutoCAD *Boundary* command is used to find the perimeter. Generally this command willonly work on closed or overlapping objexts. Use*Area by Lines & Arcs* for other applications.

Prompts

Pick point inside area perimeter: *pick a point*

Pick area label centering point: *(pick point)*The area is then plotted at the point selected.

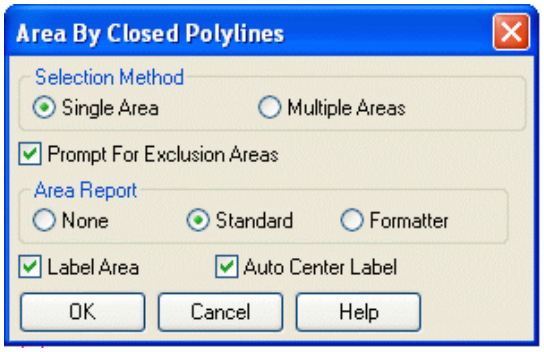
Prerequisite: Set Area Label Defaults.

Keyboard Command: ptarea

Area by Closed Polylines

Function

This command will calculate and report the area of multiple polylines.



A typical report is shown below.

Polyline Area 05/19/2002 13:56

Polyline Area: 9392.8 sq meters

Polyline Perimeter: 572.17 meters

Polyline Area: 27808.4 sq meters

Polyline Perimeter: 847.85 meters

Total Area: 37201.2 meters

Total Perimeter: 1420.02 meters

Prerequisite: Set Area Label Defaults.

Keyboard Command: plarea

Area Descriptions

Function

This command will allow you to name an area defined by a closed polyline.

Prerequisite: A closed polyline

Keyboard Command: tag_area_desc

Function

This command will allow you to pick on polylines or search your entire drawing to find descriptions.

Prerequisite: None

Keyboard Command: id_area_desc

Function

This command will remove any area descriptions on the polylines you select.

Prerequisite: Tag Area Descriptions

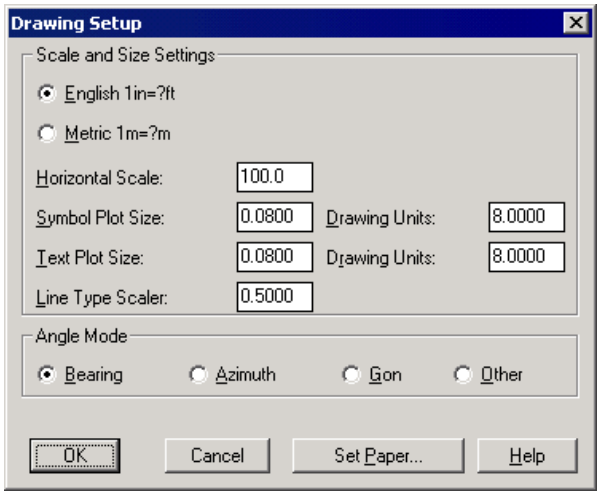
Keyboard Command: untag_area_desc

Settings Menu 11

Drawing Setup

Function

This command displays a dialog box for setting drawing parameters, including the plotting scale, size of symbols, label annotation size, and the drawing mode.



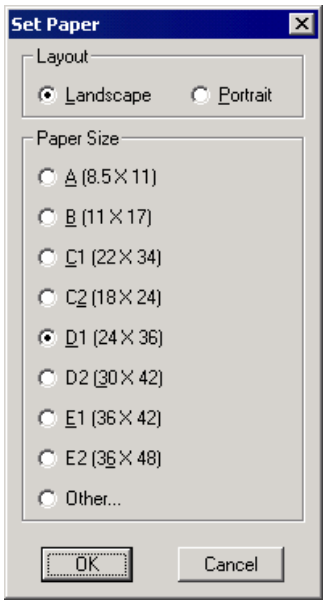
1 Under Scale and Size Settings you can determine scale and size of drawing entities.

- **English 1in=?ft**: This option tells TakeOff which unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot. In metric, one unit equals one meter.
- **Metric 1m=?m**: This option sets the metric scale to meters only.
- **Horizontal Scale**: This option allows you to set the horizontal scale of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' in your drawing.
- **Symbol Plot Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Symbol Plot Size is not entered in Drawing Units.
- **Drawing Units**: This field shows the result of the Symbol Plot Size value (the scaler) multiplied by the horizontal scale.
- **Text Plot Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units.
- **Drawing Units**: This field shows the result of the Text Plot Size value (the scaler) multiplied by the horizontal scale.
- **Line Type Scaler**: This option sets the linetype scale by multiplying this scaler by the horizontal scale.

2 Under Angle Mode you determine how angles are entered and displayed.

- **Bearing**: This option sets reporting to bearing mode for any of the inquiry commands.
- **Azimuth**: This option sets reporting to north based azimuth mode for any of the inquiry commands.
- **Gon**: This option sets reporting to gon mode for any of the inquiry commands.

3 The Set Paper button allows you to draw a rectangle on the screen that represents the edge of your paper. After you have set the horizontal scale, press the Set Paper button and the following dialog box appears



.

- **Layout**: This option lets you specify landscape or portrait paper orientation. Landscape layout is where the width of the page is greater than the height of the page. Portrait layout is the opposite.

- **Paper Size**: This option allows you to specify the paper size. The numbers in parenthesis represent drawing units and will be multiplied by the horizontal scale to determine the rectangle to be drawn. If you select the Other option, you will be prompted on the command line for the horizontal and vertical sizes of the paper.

Prompts

1 Pick or Type lower left corner point for border <(5000.00 5000.00 0.0)>: **pick point**

2 Erase existing Set Paper boundary [<Yes>/No]? **Y**

This prompt only appears if there is an existing paper boundary in this drawing

3 Set Limits [Yes/<No>]? **Y**

If you answer Yes to Set Limits, drawing limits are enabled, and TakeOff restricts the coordinates you can enter to within the paper boundary. Drawing limits also determines the area of the drawing that can display grid dots, and the minimum area displayed by the Zoom All command on the View menu. To turn drawing limits off, type in LIMITS on the command line and set to Off.

Prerequisite: None

Keyboard Command: SETUP

Preferences

Function

This command allows the user to set up a number of options, pertaining to the directory file system, color combinations for the display screen, text fonts, workspace utitlities like shortcut keys etc.

Prerequisites: None

Keyboard Command: _preferences

Configure

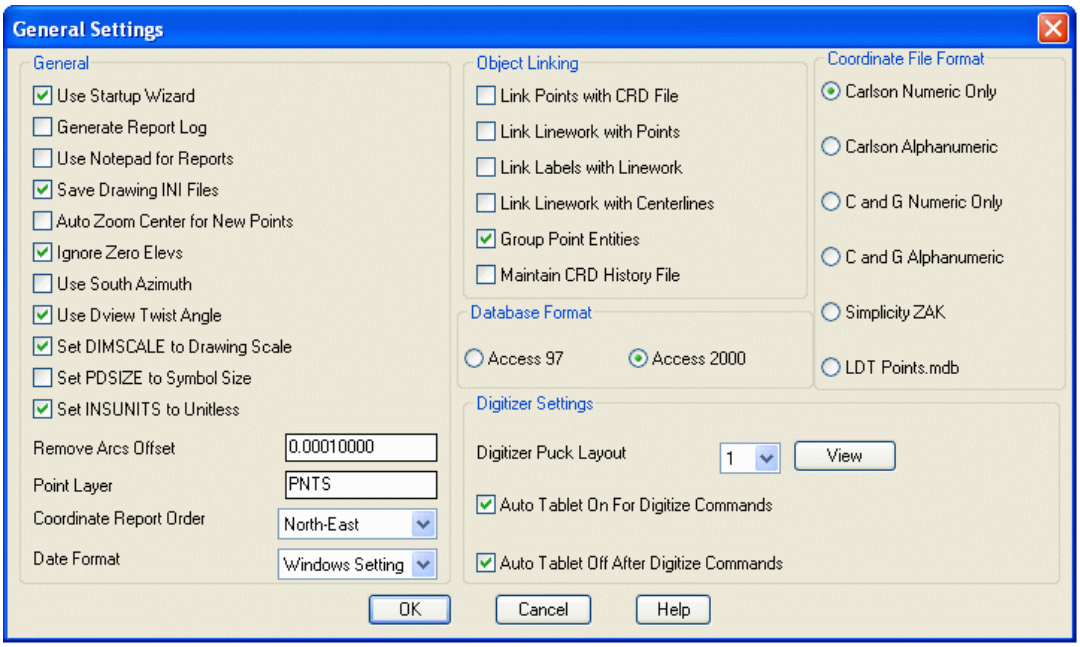
Function

This command allows you to set the default settings that are used each time you start a new drawing or load an existing drawing. These settings are stored in *.ini files in the Carlson TakeOff directory. Configure restores the current drawing settings to these default settings.

1 In the Configure dialog box you choose between General Settings, Takeoff Module, Drawing Setup, and AutoCAD Settings.



2 In the General Settings dialog box you can set options for Carlson Takeoff.



A Under General, you can choose options relevant to points, angles, and start up.

- **Use Startup Wizard**: This option controls whether the wizard appears when you create a new drawing.
- **Generate Report Log**: This option allows output from several commands to be accumulated in a report buffer. Any report that is displayed in the standard report viewer is also added to the report log. While activated, the report log resides in the lower left corner of the desktop as a minimized title bar displaying the number of lines in the report

buffer. To view the report log, pick the maximize icon on this title bar. You can edit the report log, save it to a file, or print it. To quickly turn the report log on and off, you can type REPORT at the command prompt, which toggles the report log on/off.

- **Use Notepad for Reports**: When this toggle is turned on, whenever a report is generated, it will appear in a Windows Notepad instead of the TakeOff Report Viewer.
- **Save Drawing INI Files**: This option creates an .ini file with the same name as the .dwg file to store the project data files for the drawing.
- **Put Data Files in DWG Directory**: This option sets the Data Path to the directory of the drawing. The Data Path is the default directory for data files such as the coordinate file (.crd).
- **Auto Zoom Center for New Points**: This option will zoom center on new points.
- **Ignore Zero Elevs**: This option causes entities with zero elevations to be excluded from calculations, etc.
- **Use South Azimuth**: This option allows you to use a south azimuth for calculations.
- **Use Dview Twist Angle**: This option keeps text horizontal to a twist screen view.
- **Set Dimscale to Drawing Scale**: This option sets the AutoCAD Dimscale setting to equal the current drawing scale.
- **Set PDSIZE to Symbol Size**: When checked, the system variable PDSIZE will be set to the same size as the symbol size that you set in Drawing Setup. PDSIZE controls the display size of AutoCAD point entities. Normally AutoCAD point entities are displayed as a dot, and the size does not apply. You may modify the point display type by changing the system variable PDMODE. For example, if you set PDMODE to 64, point entities are displayed as a square regardless of the TakeOff symbol type used.
- **Remove Arcs Offset:** This makes arcs into straight lines if the arcs are smaller than the offset.

- **Point Layer**: You can assign a default layer name for points.
- **Date Format**: You can control the display of dates in Carlson TakeOff reports with this popdown menu. The default is 'Windows Setting' which allows you to control it with Windows Control Panel. Several other common formats are available.



- **Digitizer Puck Layout**: This option sets the layout for the digitizer puck keys. Pick the View button to see a graphic of the different layouts.

B Under Support Paths, you must determine paths for file allocation and retrieval.

- **MS Excel Path**: You determine the path for the ∗.exe file for MS Excel is located.
- **Coordinate Report Order**: You can choose whether coordinates are reported in northing-easting or easting-northing order.

C Under Object Linking, you can set reactors to the drawing entities.

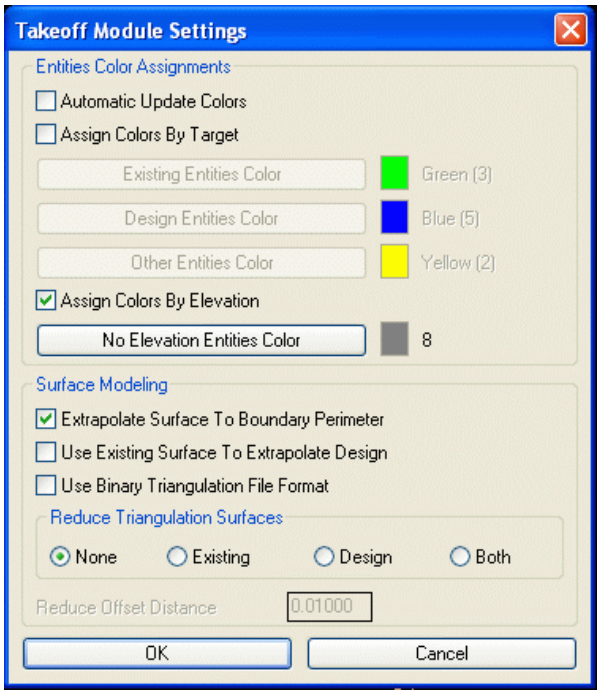
- **Link Points with CRD File**: This option attaches a reactor to the point entities so that any change to the entities such as MOVE or ROTATE will update the coordinates in the coordinate file.
- **Link Linework with Points**: This option attaches reactors to line and polyline entities that are drawn by point number so that moving the points automatically moves the linework.
- **Link Labels with Linework**: This option applies to bearing/distance annotation. This link updates the annotation when the linework is modified.
- **Group Point Entities**: This option joins the three entities of a point (attribute block, symbol, node). For each point, selecting any one of these entities selects all three entities for the point.

D Under CRD File Pt# Format, you can set the point number format for coordinate files.

- **Carlson Numeric Only**: This is the default format upon installation. Point numbers cannot contain letters and must be in the range from 1 to 32767.
- **Carlson Alphanumeric**: This native Carlson format allows letters in the point numbers, and the point name can be up to 10 characters. Any combination of letters and numbers is acceptable.
- **C&G Numeric**: This format of the C&G division supports up to 5 digits, with a 65000 point limit.
- **C&G Alphanumeric**: This format of the C&G division supports up to 10 characters, with no limit to the number of points.
- **Simplicity ZAK**: This format is the one used by Simplicity Systems programs. The file uses a Microsoft Access database format.
- **LDD Points.mdb**: This is a Microsoft Access database used by Autodesk Land Desktop. The file is typically named "points.mdb" and is found in a projects \Cogo directory. The number limitation is established by the database structure, but is frequently numeric and allows unlimited point numbers.

E Under Database Format you choose between Microsoft Access 97 or 2000 format. This only applies to new .MDB files created by Takeoff.

3 Takeoff Settings



- **Automatic Update Colors:** This refreshes colors in your drawing as they change: i.e. elevating entities, setting layers for different Targets, etc. If your drawing is very large and is slow to automatically refresh you may want to toggle this off and use the **Update Colors For Set Elevations** command under View when you want/need to see the color changes.

- **Assign Colors By Target**: This option allows you to set the Existing, Design, and Other layers to any color you define.

- **Assign Colors By Elevation**: This option allows you to set the color for entities needing elevations.

- **Extrapolate Surface To Boundary Perimeter**: When this is check ON surfaces are extended and volumes are calculated out to your boundary perimeter. When this is checked OFF surfaces and calculations end at the extents of your design data.

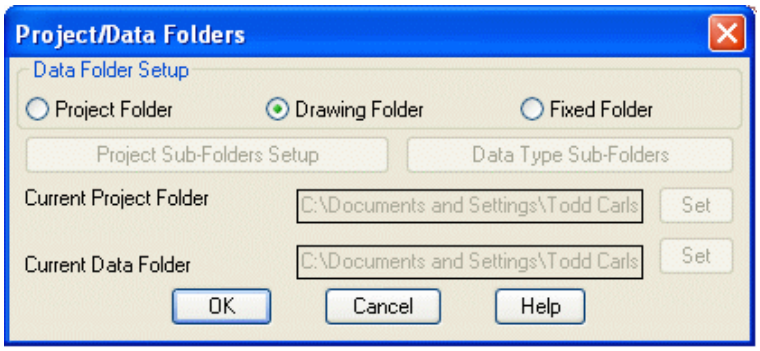
- **Use Existing Surface To Extrapolate Design**: When this is checked ON surfaces and volumes are calculated to the extents of your existing data.

- **Use Binary Triangulation File Format**: This option sets the format for the surface model files as either binary or ASCII. The binary format has a .tin file name extension and loads about twice as fast and has about 50% less file size than ASCII. The ASCII format has a .flt extension and is the legacy format used by other Carlson products and Softdesk.

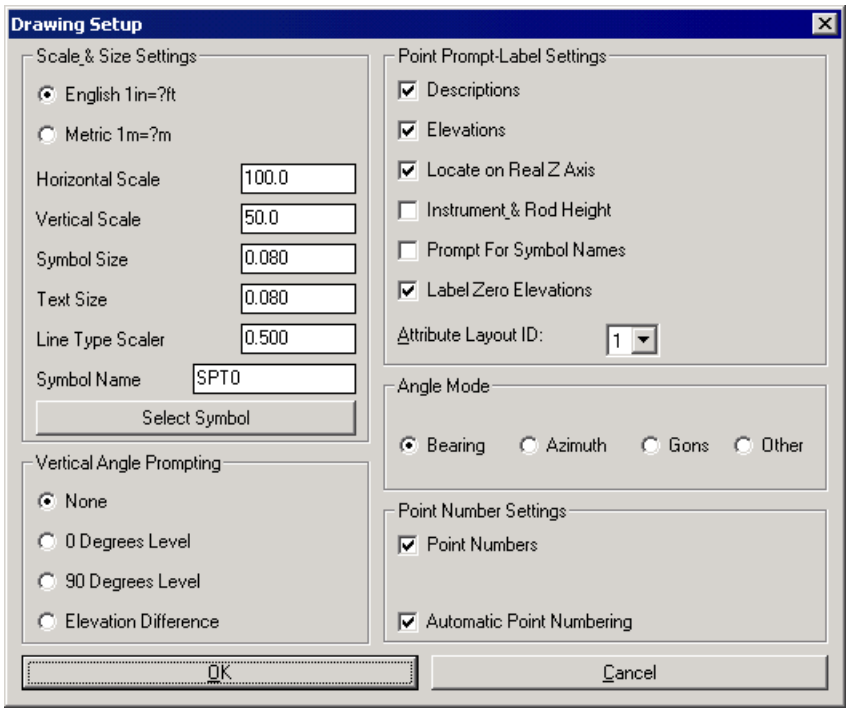
- **Reduce Triangulation Surfaces**: This causes edges within the selected surface Tin mesh to be collapsed to reduce the number of triangles, edges, and points within the mesh while having a minimal impact on the overall shape of the mesh.

Reduction Offset Distance: This setting is used by the Reduce Triangulation Surfaces command to set the reduction tolerance. Specify the maximum average distance that any point can be moved outside of the plane of any triangle that connects to that point. Values might range from .01 to .1 for most purposes.

4 This dialog box allows you to set up your. **Data Path**. You determine can where data files (*.crd, *.tin, etc.) are stored. Either in defined folders (Project Folder), the same place where your drawing (.dwg) is stored, which is the default and recommended (Drawing Folder), or in the same folder no matter which drawing you are working on (Fixed Folder).



5 In the Drawing Setup dialog box you have options for setting drawing parameters, including the plotting scale, size of symbols, label annotation size, and the drawing mode.



A Under Scale and Size Settings, you can determine scale and size of drawing entities.

- **English 1in=?ft**: This option tells the program which unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot.
- **Metric 1m=?m**: This option sets the metric scale to meters only.
- **Horizontal Scale**: This option allows you to set the horizontal scale of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' in your drawing.
- **Vertical Scale**: This option allows you to set the vertical scale of the drawing.
- **Symbol Size**: . This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Symbol Plot Size is not entered in Drawing Units
- **Text Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units.
- **Line Type Scaler**: This option sets the linetype scale by multiplying this scaler by the horizontal scale.
- **Symbol Name**: This option allows you to set the default symbol name for points.
- **Select Symbol**: Click this button to graphically select the default symbol.

B Under Point Prompt-Label Settings, select the options that determine how the points are to be labeled and how you will be prompted for point entry.

- **Descriptions**: Determines whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.
- **Elevations**: Sets prompting and labeling for point elevations.
- **Locate on Real Z Axis**: Switches between locating points at zero elevation and at the actual stored elevations.
- **Instrument & Rod Height**: Turns on prompting for instrument and rod heights when creating points.

• **Prompt for Symbol Name**: When checked, the program will prompt for a symbol name as each point is drawn. Otherwise, the default symbol name set in this dialog box will be used.

• **Attribute Layout ID**: Controls the location of the point number, elevation, and description. These attribute layouts are defined in drawings that are stored in the TakeOff Support directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then edit the associated SRVPNO drawing.

C Under Angle Mode, you determine how angles are entered and displayed.

• **Bearing**: This option sets reporting to bearing mode for any of the Inquiry commands.

• **Azimuth**: This option sets reporting to north based azimuth mode for any of the Inquiry commands.

• **Gon**: This option sets reporting to gon mode for any of the Inquiry commands.

• **Other**: Allows you to set a custom angle mode by using the Units Control command (described later in this chapter).

D Under Vertical Angle Mode select an option to determine how the vertical angle is calculated. Vertical Angle Prompt applies to creating points with commands such as Traverse.

• **None**: The vertical angle will not be used to calculate point elevations.

• **0 Degrees Level**: The vertical angle is used to calculate elevation and horizontal distance.

• **90 Degrees Level**: The zenith angle is used to calculate elevation and horizontal distance.

• **Elevation Difference**: Use the elevation difference to calculate the elevation.

E Under Point Number Settings, select the options that determine whether you will be prompted for point numbers by the commands that locate points.

• **Point Numbers**: When this toggle is checked on, points that are inserted by TakeOff commands are shown with a point number, and a coordinate is stored in the current coordinate (.crd) file. When this toggle is off, points are shown with no point number plotted and no coordinate is stored in the current coordinate (.CRD) file.

• **Automatic Point Numbering**: If this toggle is checked on, commands that locate a point will automatically insert a point number for each point drawn on the screen. If Automatic Point Numbering toggle is off, commands that locate a point will prompt for a point number.

5 AutoCAD Settings: This command allows the user to set up a number of options, pertaining to the directory file system, color combinations for the display screen, text fonts, workspace utitlities like shortcut keys etc. **Keyboard Command**: _preferences

Prerequisites: None

Keyboard Command: CONFIG_SCAD

Edit Symbol Library

Function

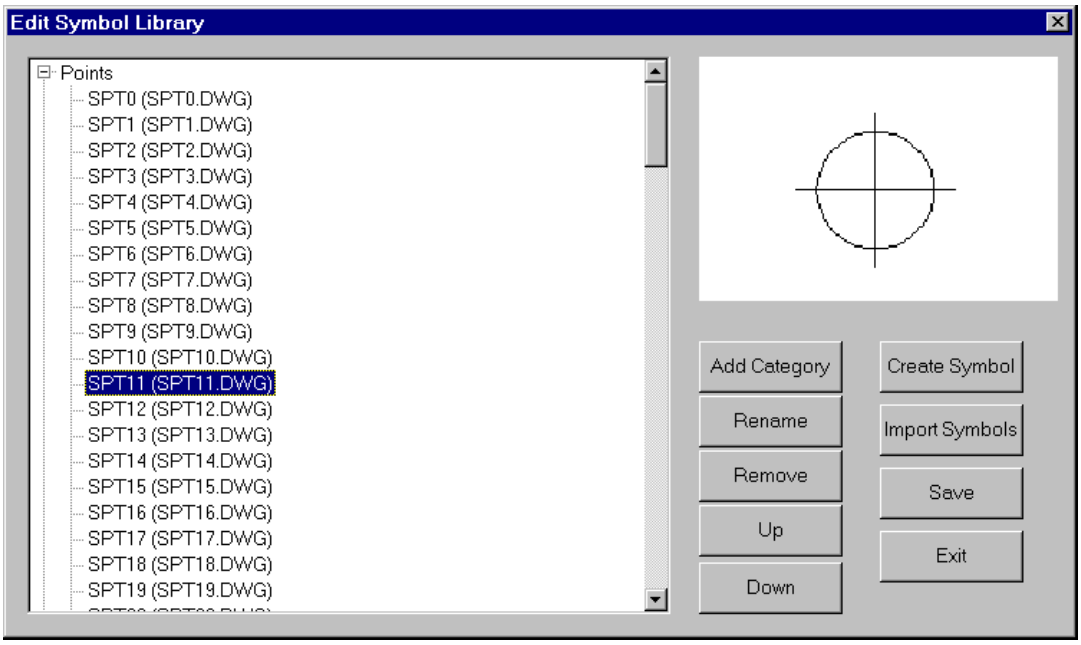
This command allows you to customize the symbol library including the symbol drawing files, symbol categories and symbol names.

Categories are a way for grouping symbols by type for your own convenience in symbol selection. A new category is added by clicking on the "Add Category" button. An edit field then appears in the tree view on the left and waits for you to enter the category name. The input is finished by pressing the Enter key.

The category may be populated by creating a new symbol from selected entities in the drawing, by importing symbol drawing files on the your computer or by moving existing symbols from one category to another.

To create a new symbol, open a drawing which has the entities to be used in the symbol. The symbol should be drawn at unit size (scale 1:1) because Carlson will scale the symbol by the current drawing scale when the symbol is used. Highlight the category for the symbol and click on the "Create Symbol" button. A dialog appears for entering the new symbol name. Next specify the file name for the symbol. The file name has a .DWG extension and would usually reside in the Carlson SUP directory, but you may use another path. Then the program will prompt you to select the entities from the drawing for the symbol. An insertion point for the symbol must also be picked.

The "Import symbols" button brings up a file selection dialog which allows you to select multiple files to be added to the current category (to select multiple files use Shift or Control keys along with the mouse). If the files being imported are not in SUP directory, the program will offer an option of copying them there or referring to the current location.



By default the symbol description is the same as file name. The description for the symbol or category name may be changed by highlighting that name and clicking on "Rename" button, the name being edited is then placed into edit mode

To move a symbol into a different category, select the symbol to be moved on the tree and click an "Up" or "Down" button as many times a needed to reach the desired category.

The symbols are sorted alphabetically within each category, while categories are remaining in the order placed to allow the more frequently accessed categories be on top.

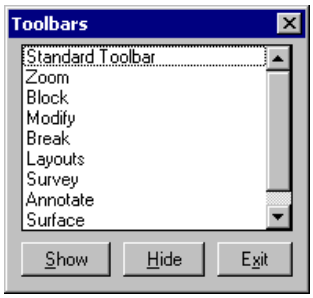
Prerequisites: Symbols

Keyboard Command: editptsym

Toolbars

Function

This command allows you to display and hide toolbars. Click on a toolbar name and press the Show or Hide button.



- **Show**: Turns on the selected toolbar. If the toolbar is already visible, then this does nothing.
- **Hide**: Turns off the selected toolbar. If the toolbars is already hidden, then this does nothing. If the toolbar is floating, you can also turn it off by clicking the x in the upper right corner.
- **Exit**: Exits this command

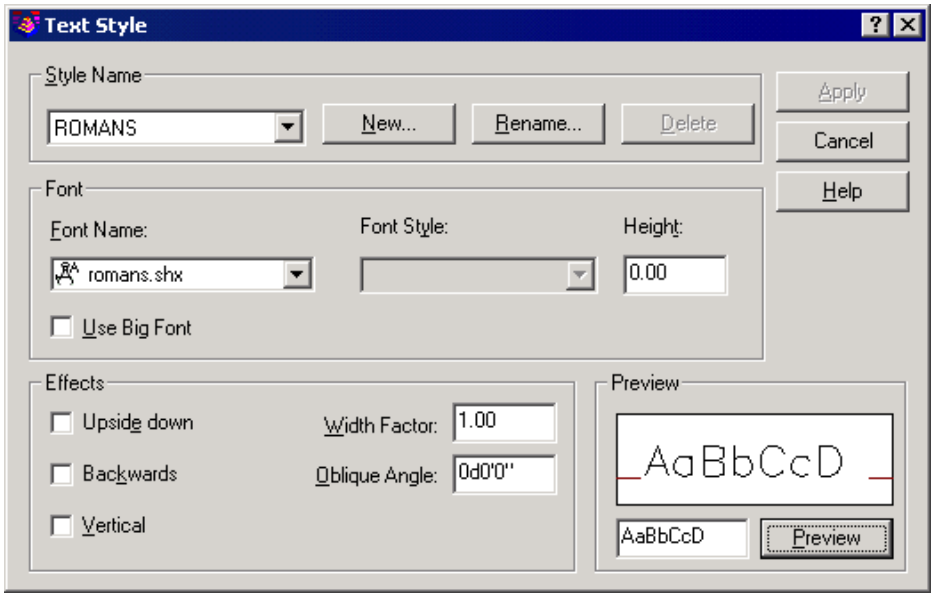
Prerequisite: None

Keyboard Command: TBARCFG

Text Style

Function

This command creates or modifies named styles and sets the current style for text in your drawing



1 Under Style Name, you can display text style names, add new styles, and rename and delete existing styles. The list contains defined style names and displays the current style by default. To change the current style, select another style from the list, or choose New to create a new style.

- **New**: This option displays the New Text Style dialog box and automatically supplies the name "stylen" (where n is the number of the supplied style) for the current settings. You can accept the default or enter a name and choose

OK to apply the current style settings to the new style name.

- **Rename**: This option displays the Rename Text Style dialog box. The text style listed is renamed when you enter a new name and choose OK.
- **Delete**: This option deletes a text style. Select a name from the list to make it current, and then choose Delete.

2 Under Font you can change the style's font.

- **Font Name**: This field lists the font family name for all registered TrueType fonts and all compiled shape (SHX) fonts in the TakeOff Fonts directory. When you select a name from the list, the program reads the file for the specified font. The file's character definitions are loaded automatically unless the file is already in use by another text style. You can define several styles that use the same font.
- **Font Style**: This field specifies font character formatting, such as italic, bold, or regular. When Use Big Font is selected, this option changes to Big Font Name and is used to select a Big Font file name.
- **Height**: This field sets the text height based on the value you enter. If you enter 0.0, the program prompts for the text height each time you enter text using this style. Entering a height greater than 0.0 sets the text height for this style. TrueType fonts can be displayed at a smaller height than SHX fonts with the same height setting. The text height you specify may not be accurately represented by uppercase letters in TrueType fonts supplied with TakeOff.
- **Use Big Font**: This option specifies an Asian-language Big Font file. Use Big Font is available only if you specify an SHX file under Font Name. Only SHX files are valid file types for creating Big Fonts.

3 Under Effects, you modify characteristics of the font, such as its height, width factor, and obliquing angle and whether it is displayed upside down, backwards, or vertically aligned. TrueType fonts using the effects described in this section might appear bold on the screen. Onscreen appearance has no effect on plotted output. Fonts are plotted as specified by applied character formatting.

- **Upside Down**: This option displays the characters upside down.
- **Backwards**: This option displays the characters backwards.
- **Vertical**: This option displays the characters aligned vertically. Vertical is available only if the selected font supports dual orientation. Vertical orientation is not available for TrueType fonts.
- **Width Factor**: This option sets the character spacing. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.
- **Oblique Angle**: This option sets the obliquing angle of the text. Entering a value between -85 and 85 makes the text oblique.

4 Under Preview, you can display sample text that changes dynamically as you change fonts and modify the effects. To change the sample text, enter characters in the box below the character preview image.

- **Preview**: This field updates the sample text in the character preview image according to any changes you've made in the dialog box. Height has no effect in the character preview image because a very large text height might show little or no text.

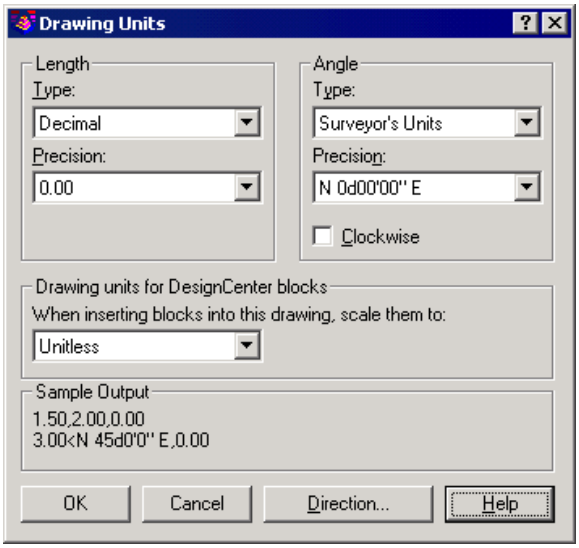
Prerequisite: None

Keyboard Command: STYLE

Units Control

Function

The Drawing Units dialog box controls coordinate and angle display formats and determines precision.



1 Under Length, you specify the current unit of measurement and the precision for the current units.

- **Type**: This field sets the current format for units of measure. The values include Architectural, Decimal, Engineering, Fractional, and Scientific. The Engineering and Architectural formats produce feet-and-inches displays and assume that each drawing unit represents one inch. The other formats can represent any real-world unit.
- **Precision**: This field sets the number of decimal places for the current units display.

2 Under Angle you specify the current angle format and the precision for the current angle display.

- **Type**: This field sets the current angle format.
- **Precision**: This field sets the precision for the current angle display.

TakeOff uses the following conventions for the various angle measures: decimal degrees appear as decimal numbers, grads appear with a lowercase g suffix, and radians appear with a lowercase r suffix. The degrees/minutes/seconds format uses d for degrees, ' for minutes, and " for seconds, for example:

123d45'56.7"

Surveyor's units show angles as bearings, using N or S for north or south, degrees/minutes/seconds for how far east or west the angle is from direct north or south, and E or W for east or west, for example:

N 45d0'0" E

The angle is always less than 90 degrees and is displayed in the degrees/minutes/seconds format. If the angle is precisely north, south, east, or west, only the single letter representing the compass point is displayed.

- **Clockwise**: This option calculates positive angles in the clockwise direction. The default direction for positive angles is counterclockwise.

When the program prompts for an angle, you can point in the desired direction or enter an angle regardless of the setting specified for Clockwise.

3 Under Drawing Units for TakeOff DesignCenter blocks, you can control the unit of measurement used for block insertions. A block created in units that differ from the units specified in this option is scaled and inserted in the specified units. Select Unitless to insert the block as is and not scale the block to match the specified units. Source content units and Target drawing units settings in the User Preferences tab of the Options dialog box under the Settings menu are used when Insert Units are not defined.

4 Sample Output displays an example of the current settings for units and angles.

Direction displays the Direction Control dialog box described below.



A The Base Angle determines where 0 degrees is located when the program calculates angles. The base angle sets the direction of the base angle. These options affect the entry of angles, object rotation angles, the display format, and the entry of polar, cylindrical, and spherical coordinates. Choose East, North, West, or South, or choose Other to indicate an alternative direction. The default direction for the zero angle is East. In TakeOff, the base angle is relative to the orientation of the user coordinate system.

- **East**: Sets the base angle to east (default is zero degrees).
- **North**: Sets the base angle to 90 degrees north.
- **West**: Sets the base angle to 180 degrees west.
- **South**: Sets the base angle to 270 degrees south.
- **Other**: Sets a direction different from the points of the compass.
- **Angle**: Sets the angle. Available only when Other is selected.
- **Pick an Angle**: Uses the pointing device to define the angle based on the angle of an imaginary line connecting any two points you specify. Available only when Other is selected.

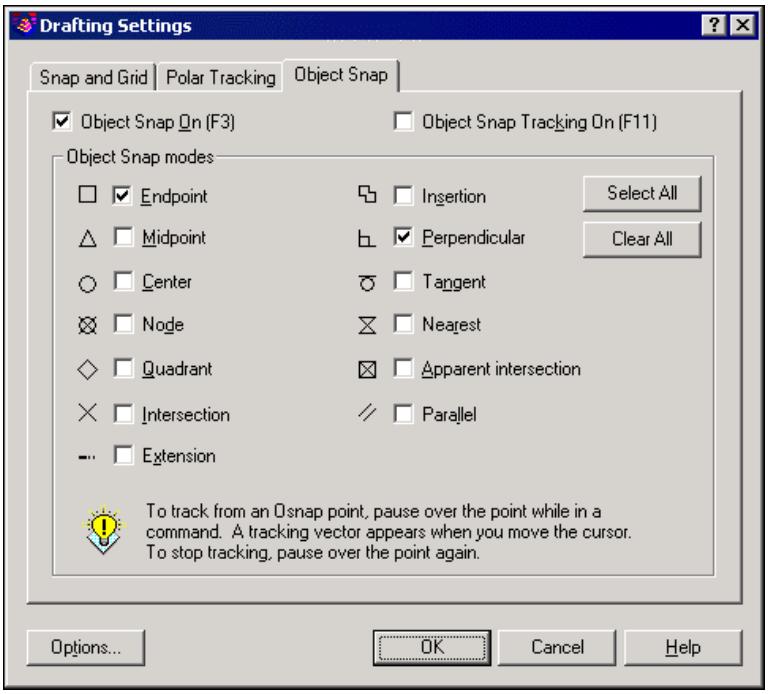
Prerequisite: None

Keyboard Command: UNITS

Object Snap

Function

The Drafting Settings dialog box sets object snap modes.



1 Under Object Snap, you set object snaps.

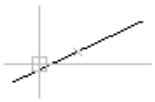
- **Object Snap On**: This option turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. This setting is also controlled by the OSMODE system variable.

- **Object Snap Tracking On**: This option turns object snap tracking on and off. With object snap tracking the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps.

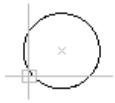
2 Under Object Snap Modes, you turn on running object snaps.

- **Endpoint**: Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray or to the closest corner of a trace, solid, or 3D face.

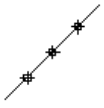
- **Midpoint**: Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, solid, spline, or xline.



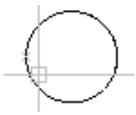
- **Center**: Snaps to the center of an arc, circle, ellipse, or elliptical arc.



- **Node**: Snaps to a point object.

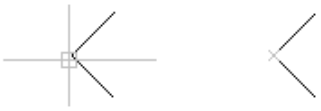


- **Quadrant**: Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.



• **Intersection**: Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline. Intersection snaps to the edges of regions and curves, but does not snap to the edges or corners of 3D solids.

Extended Intersection snaps to the imaginary intersection of two objects that would intersect if the objects were extended along their natural paths. Carlson TakeOff automatically turns on Extended Intersection when you select the Intersection object snap mode. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Intersection and Extended Intersection work with edges of regions and curves, but not with edges or corners of 3D solids.



• **Extension**: Causes a temporary extension line to display when you pass the cursor over the endpoint of objects, so you can draw objects to and from points on the extension line.

• **Insertion**: Snaps to the insertion point of an attribute, a block, a shape, or text.

• **Perpendicular**: Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, solid, spline, or xline. Carlson TakeOff automatically turns on Deferred Perpendicular snap mode when the object you are drawing requires you to complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, the program displays a Snaptip and marker.



• **Tangent**: Snaps to the tangent of an arc, circle, ellipse, or elliptical arc. Carlson TakeOff automatically turns on Deferred Tangent snap mode when the object you are drawing requires you to complete more than one tangent snap. For example, you can use Deferred Tangent to draw a line that is tangent to two arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, the program displays a marker and Snaptip. If you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.



• **Nearest**: Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, spline, or xline.

• **Apparent Intersection**: Apparent Intersection includes two separate snap modes: Apparent Intersection and Extended Apparent Intersection. You can also locate Intersection and Extended Intersection snap points while running Apparent Intersection object snap mode is on. Apparent Intersection snaps to the apparent intersection of two objects (arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline) that do not intersect in 3D space but may appear to intersect in the drawing display. Extended Apparent Intersection snaps to the imaginary intersection of two objects that would appear to intersect if the objects were extended along their natural paths. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Apparent and Extended Apparent Intersection work with edges of regions and curves but not with

edges or corners of 3D solids.



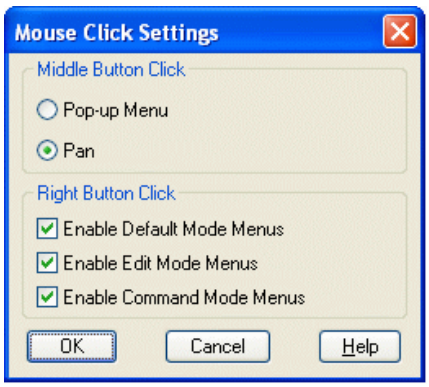
- **Parallel**: Draws a vector parallel to another object whenever Carlson TakeOff prompts you for the second point of a vector. After specifying the first point of a vector, if you move the cursor over a straight line segment of another object, the program acquires the point. When the path of the object you create is parallel to the line segment, the program displays an alignment path, which you can use to create the parallel object.
- **Clear All**: This option turns off all object snap modes.
- **Select All**: This option turns on all object snap modes.

Prerequisite: None

Keyboard Command: OSNAP

Mouse Clicking Settings

This command can be used to make custom mouse click preferences. The Middle Button Click applies to a 3-button mouse and chooses between using the middle button for real-time pan or to show an Object Snap pop-up menu. For the right mouse button, there are different levels of pop-up menus that can be activated. With all these menus off, the right button will be used like the Enter keyboard.



Pulldown Menu Location: Inquiry
Keyboard Command: CLICKSET
Prerequisite: none

Crosshairs

Function

Set X-Hairs sets the cross-hairs either to align with the selected line or polyline or to a user-specified slope.

Reset X-Hairs restores the cross-hairs alignment to horizontal.

Pull-Down Menu Location: Settings->Crosshairs

Prerequisite: Line entity.

Keyboard Command: setxhairs, resetxhairs

Set UCS to World

Function

This command sets the UCS (user coordinate system) to the world coordinate system (WCS). Carlson TakeOff works exclusively in the world coordinate system and there is no way to change this setting. In AutoCAD, it is possible to change the coordinate system from WCS. If you receive a drawing in which the coordinate system is not set to world, use this command to restore the UCS.

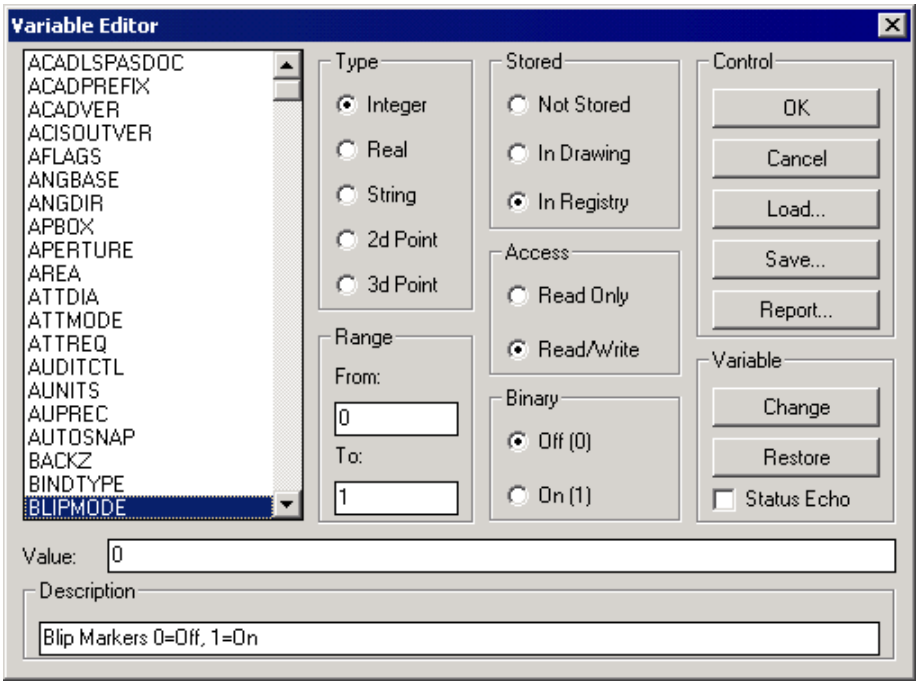
Prerequisite: None

Keyboard Command: UCS_WORLD

Set Environment Variables

Function

The AutoCAD engine stores the values for its operating environment and some of its commands in system variables. Each system variable has an associated type: integer, real, point, switch, or text string. This command allows you to list or change the values of system variables.



1 Dialog Fields

- **List Box**: Contains a list of the variables associated with the currently running version of AutoCAD. There are more items than will display on the list box, use the scroll bar to move up and down through the list. Picking on

an item in the list box makes it the current item, causing the information about the item to be displayed, and can be affected by most of the edit commands explained below.

- **Edit Field**: When an item on the list box is picked, its current setting is displayed in the edit field. If you intend to make changes in an item, use standard editing procedures including the use of arrow keys and/or pointer movements to make changes. Once changes have been made, you must use the CHANGE options explained below to effect changes. Pressing enter at the edit field will have no effect on the item in the list. If the item selected is a read-only variable, the edit field will be grayed-out and will not allow input.

- **Description**: When an item on the list box is picked, its definition is referenced and displayed in this field. This can be a benefit in learning the uses of the assorted system variables. This is a display only field, so you can't change the description given.

2 Under Type Group, the type of variable will be displayed indicated by one of the radio buttons. Each of these types are explained below for your benefit. For additional information on variable types used by AutoCAD, obtain and consult a source of AutoCAD documentation.

- **Integer**: Defined as a whole number in the range from -32767 to +32768, no decimal value accepted.

- **Real**: Defined as a real number in the range from -1.797E+308 to +1.797E+308, with extreme decimal accuracy maintained.. Some real variables have a smaller range than previously stated.

- **String**: Defined as a sequential array of characters in the range from 0 to 65535 characters, with a range of ASCII (0-255). Numbers can be included in strings, even though they have no mathematical significance.

- **2D Point**: Defined as a list of two real numbers in the range from -1.797E+308 to +1.797E+308 separated by a comma, having extreme decimal accuracy maintained. Always maintain the X,Y format, one (and only one) comma must be used, separating the X and Y.

- **3D Point**: Defined as a list of three real numbers in the range from -1.797E+308 to +1.797E+308 separated by commas. While editing a 3D point, you must always maintain the X,Y,Z format, two (no less or no more), commas must used, separating the X and Y and Z values.

3 Under Range Group, the variable displayed will usually have a range displayed. The FROM value indicating the minimum, and the TO value being the maximum value accepted.

4 Under the Store Group, depending on the type of variable, AutoCAD may store the value in the drawing or the configuration file, or it may not be stored. Each of these types are explained below for your benefit.

- **Not Stored**: Some variables, such as PLATFORM and CDATE, are not stored because they are system interdependent.

- **In Drawing**: Most variables are stored in the drawing, making the drawing format more personal than just a database of objects. This allows you to open a drawing and have it behave just as though you had never left it.

- **In Config**: These are variables that remain the same regardless of the drawing opened. APERTURE and PICKBOX are just two examples of variables stored in the configuration file.

5 Under Access Group, depending on the type of variable, AutoCAD may not allow you to make changes to it. Each of these types are explained below.

- **Read Only**: Some variables, such as PLATFORM and CDATE, are read-only and therefore cannot be changed. Read-Only variables are marked and the edit field will be grayed indicating that you can't change the variable.

- **Read/Write**: Most variables are read/write and can be changed. These variables are marked and the edit field will be active so you can change the variable.

6 Under Binary Group, depending on the type of variable, the value may be off or on, yes or no. If the variable type is not binary, this group will be grayed out entirely.

• **Off (0)**: Indicate an off condition. Some variables, such as ATTREQ, are simply on or off toggles. You may change a binary item by clicking in this group to change the variable, or changing the value in the edit field.

• **On (1)**: Indicate an on condition. Binary variables are simply on or off toggles. Their range is from 0 to 1. You may change a binary item by clicking to change the variable, or changing the value in the edit field.

7 Control Buttons - These buttons are the main controls in the use of the Variable Editor. Each buttons purpose is explained below.

• **OK**: Used to accept the changes made during the variable editing process, returning you to the command prompt with changes in effect.

• **Cancel**: Used to cancel the changes made during the variable editing process, returning you to the command prompt without the changes in effect.

• **Load**: Used to load a saved set of system variables. This allows you to create a drawing, save the system variables, open a second drawing, and load those variables into that drawing. Read-only variables are skipped.

• **Save**: Used to save the current system variables to a disk file. All system variables are stored to the file, even those that are marked as read-only.

• **Print**: Used to print the current system variables. After choosing this option, you will prompted for an output filename, then the program will proceed to write the system variables to the file. This file can be loaded into any editor or word processor, edited and printed.

8 Variable Buttons - These buttons are used to control the changes in variables, while using the Variable Editor. Each buttons purpose is explained below.

• **Change**: Used to execute the changes typed into the edit field. You must use this button, simply pressing enter will not make the change.

• **Restore**: Used to cancel the changes typed into the edit field. If you make a mistake or change your mind while making changes in the edit field, press this button to restore the edit field to the value before editing.

• **Status**: Used to determine if the program will echo the status of changes being made to the command area. If this toggle is on, any changes made from the dialog will echo the change. Also if a stream of change commands is being read from a file, and the toggle is on, the changes taking place will be displayed.

Note: This command displays many more system variables than are found in Chapter 18, which contains a list of **supported** system variables. Modification of any system variable other than the supported ones found in Chapter 18 is done at your own risk, and may result in program errors requiring a re-installation of Carlson TakeOff.

Prerequisite: None

Keyboard Command: VAREDIT

Drillhole Menu 12

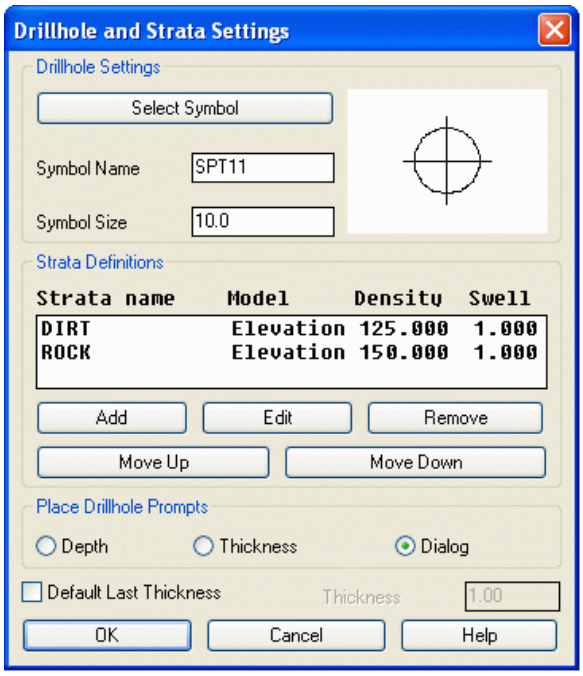
Drillhole Strata Settings

Function

This command selects drillhole symbols, defines strata, and determines how you place drillholes.

Note: The order in which the Strata are defined in the Strata Definitions list will be the default order for the strata when you create new drillholes through **Place Drillhole**.

The dialog box below shows the layout of the Drillhole and Strata Settings.



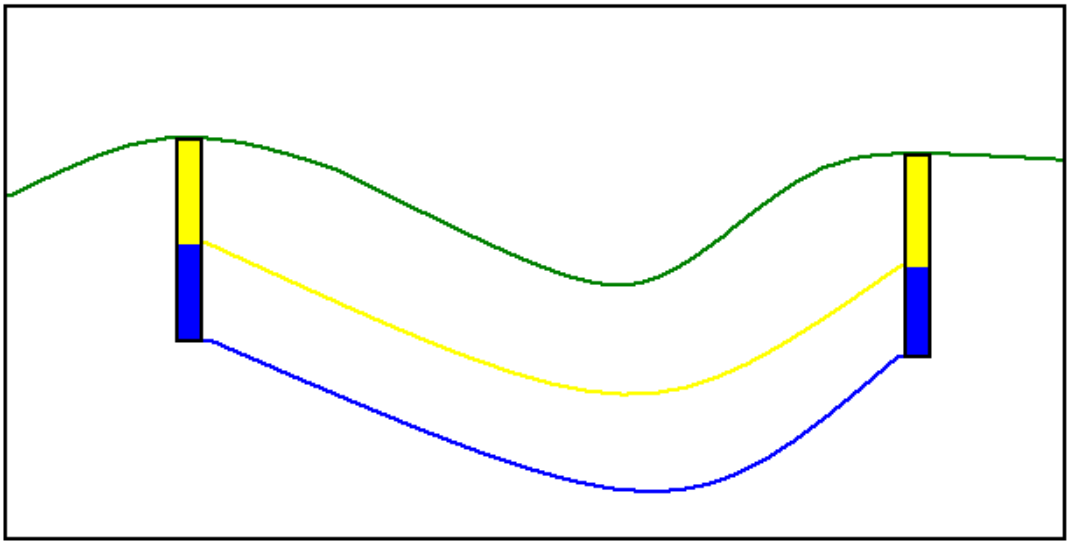
- **Select Symbol**: Select a symbol to represent the drillhole location on the screen.
- **Symbol Name:** This name corresponds to the symbol selected.
- **Symbol Size:** This field can be editted to adjust the symbols size displayed on the screen.
- **Strata Definitions:** This is not directly editable. Select the Strata you are interested by highlighting it, then select the Edit button.
- **Add:** This option adds additional strata to the available Strata name list. See Edit Strata dialog box below.

- **Edit:** Similar to Add, this option is available to make changes to the Strata, including adding a swell factor.

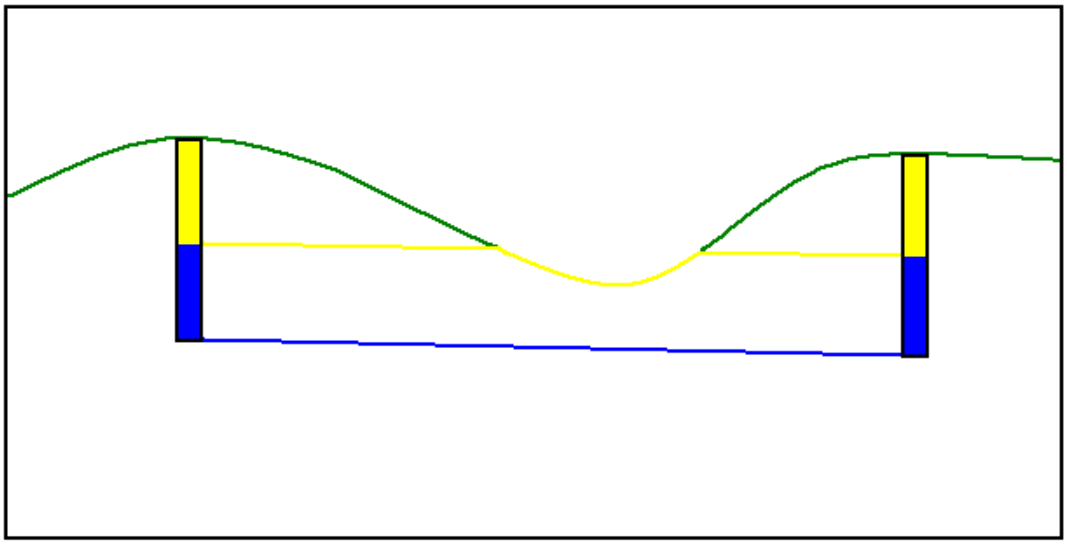
- **Strata Name:** The name of the strata.

Density (lbs/ft^3): The Strata Density field is the default density used to calculate strata tons. Density is strata-specific.

- **By Depth:** This option will generate a strata surface by modeling the strata depth values in the drill-holes. This strata surface will follow the existing ground surface at the model depth.



- **By Strata Elevation:** This option will generate a strata model that connects strata irrespective of the upper surface elevation changes.



- **Remove:** This will remove a strata name from the available strata.

- **Move Up:** This option will move the selected strata name up one place in the strata name list.

- **Move Down:** This option will move the selected strata name down one place in the strata name list.

- **By Strata Elev:** This method will generate a strata surface by modeling the strata elevation values from the drillholes. This strata surface is independent of the existing ground surface.

- **Place Drillhole Prompts:** If Depth is selected, then when you run Place Drillholes you will be prompted for the depth of each strata in your drillhole. If Thickness is selected, you will be prompted for the thickness of each strata. If Dialog is selected, you will go straight into the Place Drillhole dialog when you create a drillhole.

- **Default Last Thickness:** Will set the thickness of your bottom strata to the same amount for all your drillholes.

Prerequisite: strata information

Keyboard Command: tk_chdef

Drillhole Import

Function

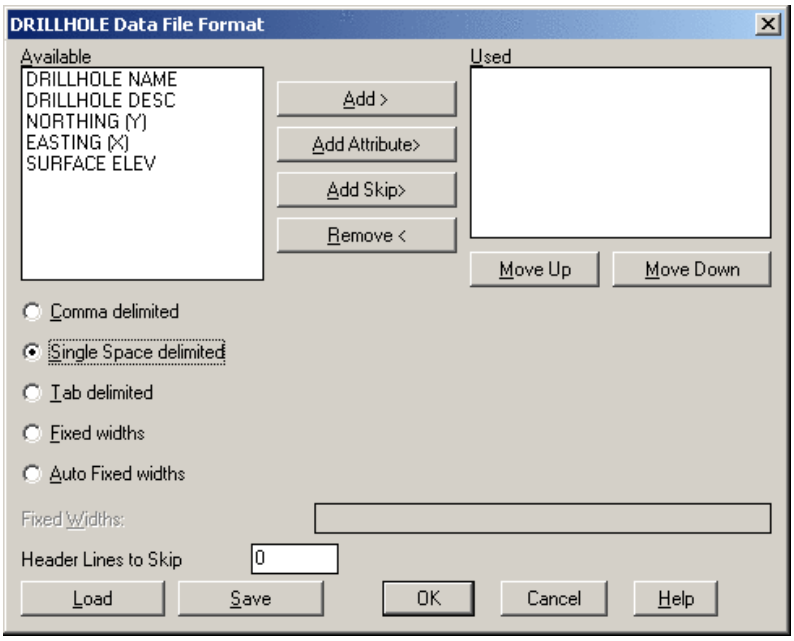
This command allows you import existing drillhole files.

When you select Drillhole Import from the Drillhole menu, a command prompt shows:

"Use seperate drillhole and strata files [Yes/<No>]?" If you have two seperate files, one with strata info,

and the other file has drillhole locations, select Yes. If you enter Yes, the dialog box below appears.

This command creates drillholes from the data contained in text files. Currently there are many company-specific formats. A Drillhole Data File Formatter that is flexible to handle almost any drillhole text file format is below. The format to use is chosen in the dialog shown here.



The import text can have comma delimited, space delimited or fixed width columns of data. All the data for a record should be on one row. For the fixed width format, choose the Fixed Width toggle and then enter the column numbers separated by spaces in the edit box. For example, "8 15 24 32".

The Custom format can import all the drillhole and strata data from one text file or the drillhole data from one file and the strata data from another file. The method to use is set at the Use separate drillhole and strata files prompt.

Use the following commmands to prepare a file format that will match the *.imp imported file.

- **Add:** Moves the selected entry from Available to Used.
- **Add Attribute:** Allows user input attributes into the Used section.
- **Add Skip:** Adds a "Skip" place holder in the Used List
- **Remove:** Moves a selected item from Used to Available list.
- **Move Up:** Moves the selected item up one place in the list.
- **Move Down:** Moves the selected item down one place in the list.
- **Comma Delimited:** Select this if your *.imp file has commas seperating each field.

- **Single space delimited:** Select this if your *.imp has a space seperating each field.

- **Tab delimited:** Select this if your *,imp file has tabs seperating each field.

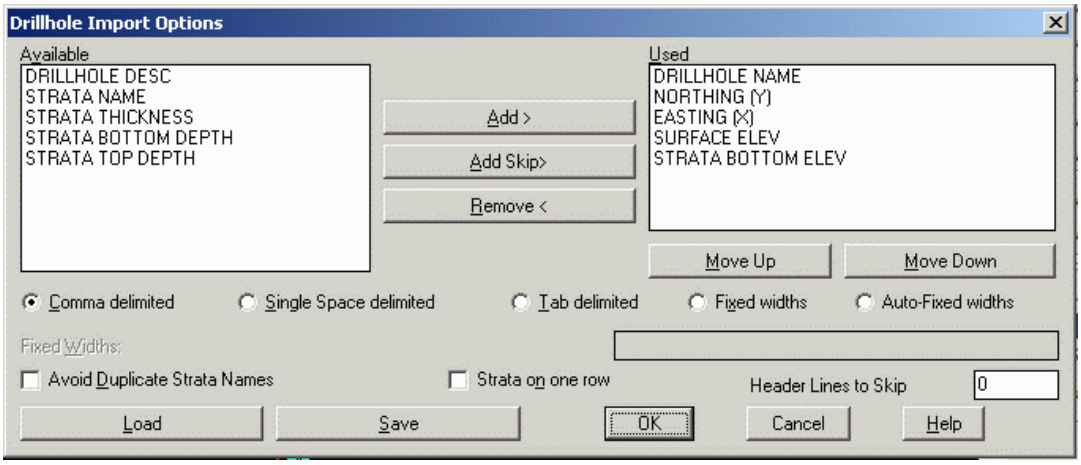
- **Fixed widths:** Select this if your ∗.imp has a defined width of space seperating each field.

- **Auto Fixed widths:** Select this to automatically determine the fixed widths that seperate each field in the ∗.imp file.

- **Header Lines to Skip:** If your ∗.imp file has header lines, enter the number of header lines here.

- **Load:** Takes you to select/brows for your ∗.imp file.
- **Save:** This command will save your imported file as a ∗.imp file.

The dialog box below details the drillhole import options.



In addition to the previously listed import commands above, this dialog box also has the following prompts:

- **Avoid Duplicate Strata Names:** Select this to prevent have more than one strata with the same name.

- **Strata on one row:** Select this option if all of your starta info is on one row.

Prerequisite: drillhole files

Keyboard Command: tk_chimport

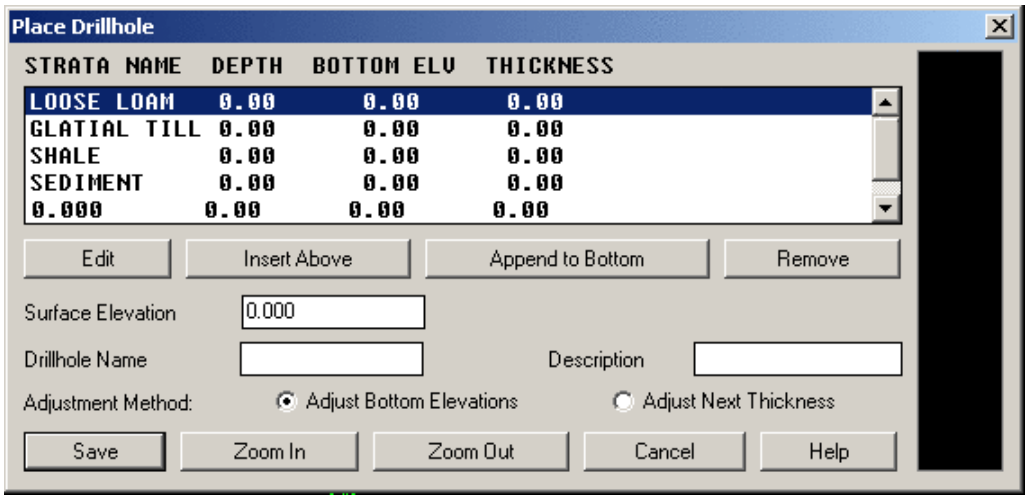
Place Drillhole

Function

This command allows you to screen pick or enter coordinates the placement of a drillhole.

Go to **Drillhole/Strata Settings**, **Place Drillhole Prompts**, to determine how you would like to be prompted. When you select **Place Drillhole** from the **Drillhole menu**, the command line prompt shows:

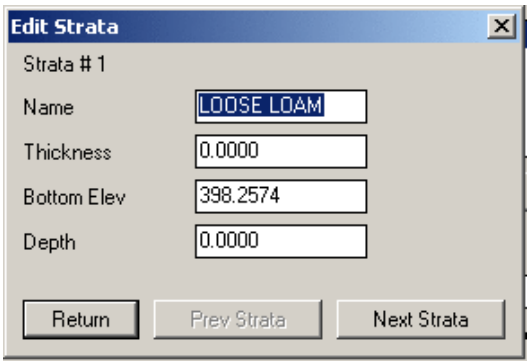
"Pick Drillhole Location:" Type in x-y coordinates or move your pointer around the screen to pick the placement of the drillhole. If you are in Dialog Mode defined in **Drillhole/Strata Settings**, once a location was picked on the screen, the following dialog box appears:



Place DrillHoles generates drillholes in the drawing that are required to run strata surface application routines. Each drillhole consists of a surface elevation, strata, and optional description(s). Every strata has a name, bottom elevation, thickness. Within a drillhole, the strata names must be unique, but each real-world strata should have the same strata name across all the drillholes. This is because strata surface applications connects together the strata with the same name.

The drillhole data can be entered in the dialog shown here, or if Depth or Thickness Mode is selected under **Drillhole/Strata Settings**, then the data can be entered in on the command line when you place eachdrillhole. Make sure to specify the surface elevation and drillhole description. While in Dialog Mode or to change data, use the Edit and Insert/append buttons to enter strata data. The symbols are defined in DrillHole/Strata Settings and drillhole may be changed in Edit DrillHole. Pick Save when done and a drillhole symbol is drawn.

- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.



When placing drillholes, every strata must be assigned a bottom elevation and a thickness. The bottom elevation is the elevation of the bottom of the strata. There are different methods for entering this information.

- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.

- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish th surface elevation of the drillhole.

- **Drillhole Name:** The name of the drillhole

- **Description:** Drillhole descriptions are intended for storing of drillhole specific information in the drillhole. One general drillhole description is predefined and user may define any number of specific drillhole descriptions. Typical additional descriptions are DRILLER, DATE, TOWNSHIP, and etc. You will be prompted for values of these descriptions in Place DrillHole.

- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.

- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.

- **Save:** This caommand saves this drillhole as listed.
- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.

- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.

- **Cancel:** Ends Drillhole placement routine without making changes.

Prerequisite: drillhole information

Keyboard Command: tk_chplace

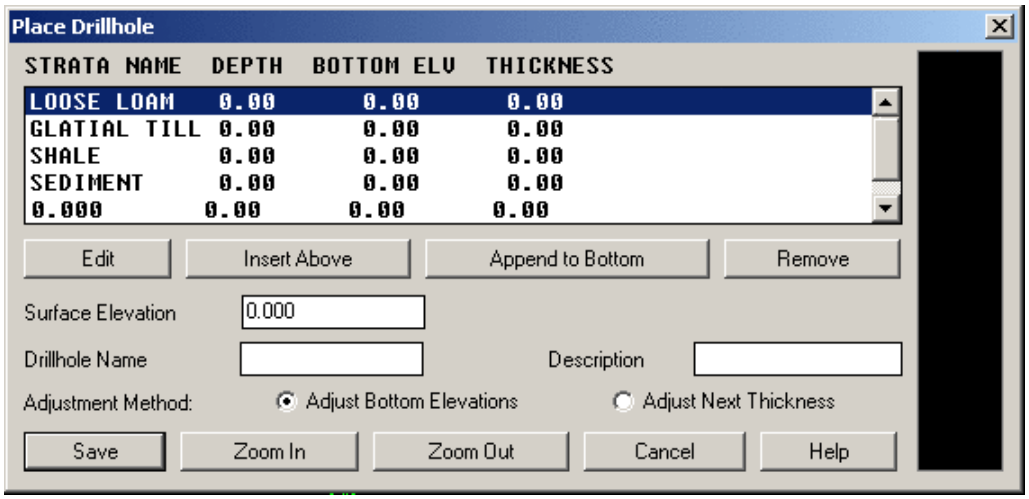
Edit Drillhole

Function

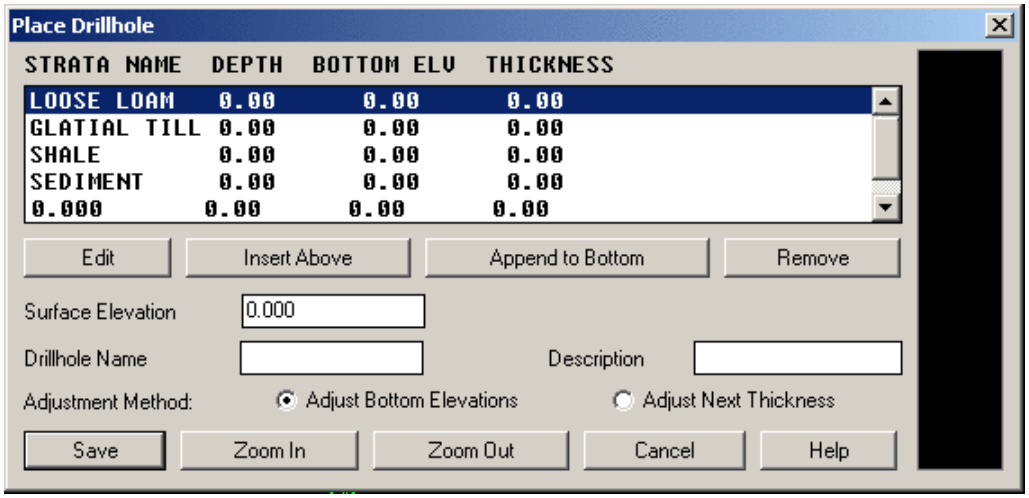
This command allows you to screen pick an existing drillhole and edit its properties.

When you select **Edit Drillhole** from the **Drillhole menu**, a command prompt shows:

"Select Drillhole to edit:" Move your pointer around the screen to pick the drillhole you want to edit. Once a drillhole is picked on the screen, the following dialog box appears:



- **Edit:** Make changes to the highlighted strata name. Thickness, Bottom Elevation, Depth.



- **Insert Above:** To add a Strata above the highlighted strata name.
- **Append to Bottom:** To add a strata to the bottom of the available strata name list.

- **Remove:** Removes a strata from the available Strata Name list.
- **Surface Elevation:** This field can be set by you to establish th surface elevation of the drillhole.

- **Drillhole Name:** The name of the drillhole
- **Description:** The screen display description of the drillhole
- **Adjust Bottom Elevations:** Will make adjustments to the bottom elevation based on thickness changes.

- **Adjust Next Thickness:** Will adjust the next thickness to hold the bottom elevation unchanged.

- **Save:** This caommand saves this drillhole as listed.
- **Zoom In:** This increases the magnification of the black view window, cross-section view of the drillhole.

- **Zoom Out:** This decreases the magnification of the black view window, cross-section view of the drillhole.

- **Cancel:** Ends Drillhole placement routine without making changes.

Prerequisite: drillhole information

Keyboard Command: tk_chedit

Reports

Function

This command allows you to generate a report of selected drillholes.

When you select **Reports** from the **Drillhole** menu, a sub-menu choice of **Standard Drillhole Report** or **Custom Drillhole Report**, is displayed.
Standard Drillhole Report

If this is selected, several prompts are asked at the command line. They are as follows:

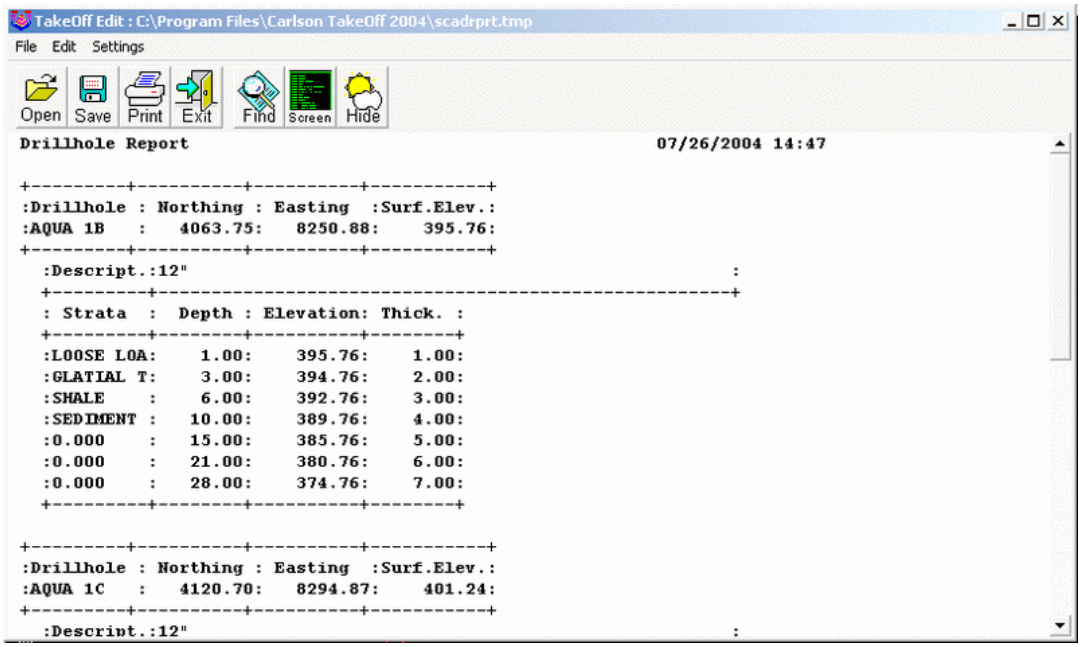
Select objects:

Add Page break between drillholes [Yes/<No>]?

Report Strata depth to [Top/<Bottom>]?

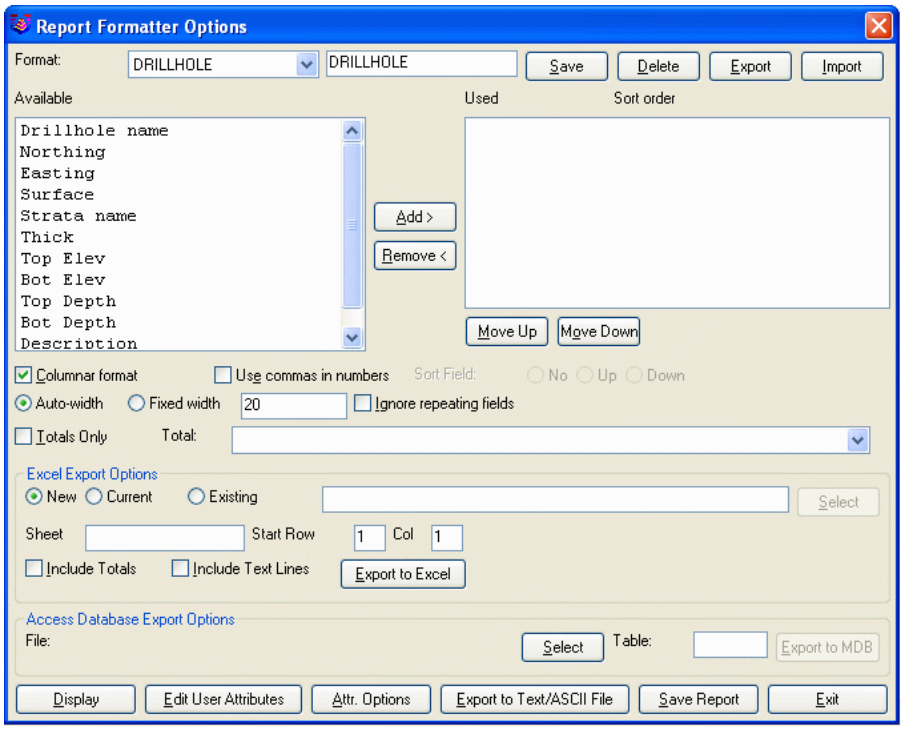
Report Strata elevation of [Top/<Bottom>]?

The report is then displayed accordingly.


TakeOff Edit : C:\Program Files\Carlson TakeOff 2004\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Drillhole Report 07/26/2004 14:47
:Drillhole : Northing : Easting :Surf.Elev.:
:AQUA 1B : 4063.75: 8250.88: 395.76:
:Descript.:12"
: Strata : Depth : Elevation: Thick. :
:LOOSE LOA: 1.00: 395.76: 1.00:
:GLATIAL T: 3.00: 394.76: 2.00:
:SHALE : 6.00: 392.76: 3.00:
:SEDIMENT : 10.00: 389.76: 4.00:
:0.000 : 15.00: 385.76: 5.00:
:0.000 : 21.00: 380.76: 6.00:
:0.000 : 28.00: 374.76: 7.00:
:Drillhole : Northing : Easting :Surf.Elev.:
:AQUA 1C : 4120.70: 8294.87: 401.24:
:Descript.:12"

Custom Drillhole Report

This function allows you to customize your report format.



Prompts:

Command: tk_chreport2
Select the Drillholes for report.
Select objects: Specify opposite corner: 271 found
262 were filtered out.

Prerequisite: drillholes

Keyboard Command: tk_chreport, tk_chreport2

Make Strata Surface

Function

This command generates multiple strata surfaces based on strata definitions and placements of drillholes. Strata surfaces are generated at the bottom of each strata. These strata surfaces can then be used in other TakeOff commands like **Calculate Total Volumes**. They can be viewed on screen, through the command **Draw Strata Surface**.
Note: By observing the command line, one can see the status of each strata surface generation.

Prerequisite: Define Drillhole/Strata Settings, Place Drillhole

Keyboard Command: tk_chgrid

Clear Strata Surface

Function

This command clears the starta surfaces previously generated with **Make Strata Surface**. This removes the strata surfaces from processing in other takeoff commands.
Note: This command will not remove the surface from the screen view. You must use the command **Erase Strata Surface** to remove them from view.
Prerequisite: Make Strata Surface

Keyboard Command: tk_chclear

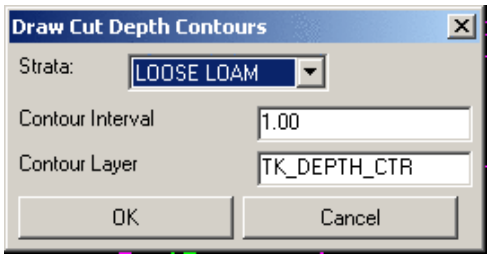
Draw Strata Cut Depth Contours

Function

This command will draw the **Strata Cut Depth Contours**. This command creates contours for the cut depth between the design surface and strata.

You must have created Strata Surfaces through the **Make Strata Surface** command.

Then select **Draw Strata Cut Depth Contours** from the **Drillhole menu**. You will be prompted to select the Strata from the dialog box below.



You can assign a contour interval and contour layer for the contours to be drawn.

Prerequisite: Make Strate Surface

Keyboard Command: tk_chdepth

Erase Strata Cut Depth Contours

Function

This command will erase the Strata Cut Depth Contours from the screen display.

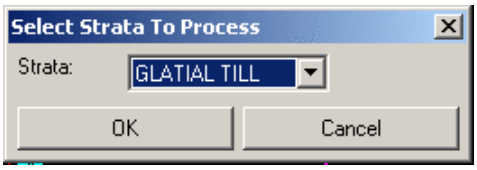
Prerequisite: Strata Cut Depth Contours

Keyboard Command: tk_chdepth2

Draw Strata Cut Color Map

Function

This command will generate a map of areas where the design surface cuts into the selected strata.



Prompts

Select point for color legend: - Use your pointing device to select the top left corner of where you want the cut color legend to be displayed.

Legend size <10.0>: - Screen display size.

Label all zones or summary [All/<Summary>]? - This pertains to the the number of elevation labels on the legend.

Prerequisite: Make Strata Surface

Keyboard Command: tk_chmap

Erase Strata Cut Color Map

Function

This command will erase all Strata Cut Color information from the screen display.

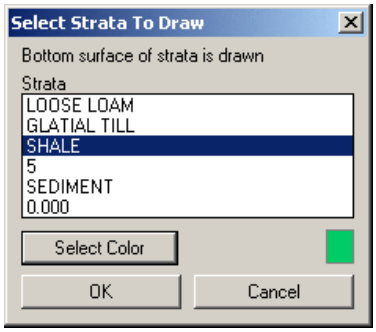
Prerequisite: Draw Strata Cut Color Map

Keyboard Command: tk_chmap2

Draw Strata Surface

Function

This command will display the selected strata surfaces as 3D faces. The bottom elevation of the strata is drawn.



A color can be selcted to distinguish each strata.

Prerequisite: Make Strata Surface

Keyboard Command: tk_chplot

Erase Strata Surface

Function

This command will erase all strata surface 3D faces from the screen display.

Prerequisite: Draw Strata Surface

Keyboard Command: tk_chplot2

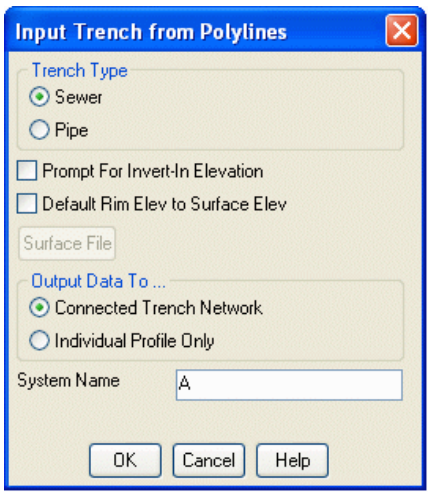
Trench Menu 13

Input Trench From Polyline

Function

This command allows you to input a trench sewer network structure from polylines. It first prompts you the **Input Trench from Polyline Dialog** where you specify the Trench Type, Trench System, and the SystemName. The Individual Profile option lets you input one trench reach at a time and save its information to a profile (.pro). The Connected Network option lets you input all the trench polylines on the drawing, merge them into a trench network structure and save the whole structure to a .sew file. For tenching or utilities without Invert-Ins, uncheck Prompt For Invert-In Elevation. If you want to set the Rim Elevation to any surface elevations, check on Default Rim Elev to Surface Elev and then use the Surface Button to select the desired .tin or .flt surface file. Click OK to start inputting trench structure.

Pick a polyline that represents a trench reach on your drawing. Takeoff extracts the coordinates of all the vertices of the polyline and prompts you for the starting station number. Takeoff would computer the station values based on the starting station number. Next, you are prompted to enter the Manhole ID (Sewer Trench) or Station ID (Pipe Trench), Invert Elevation, Manhole Elevation (Sewer Trench), and Pipe Size for every station. You can either enter the values manually or select the texts that represent these values on the drawing. When you finish inputting a polyline, the command would ask you for a profile name to store the profile data if you are doing Individual Profile; otherwise the command would ask you to pick next polyline that is in the same trench network.



Prompts:

Pick a polyline that represents a trench reach: *pick a polyline on your drawing*
Starting Station of trench reach <0.0>: *press Enter to accept 0.0 as the starting station or enter a value*

For station 0.00 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *select the invert elevation text on the drawing or enter Enter on the*

keyboard to enter the invertelevation value manually
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manholeelevation value manually*
For station 270.22 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *select the invert elevation text on the drawing or enter Enter on the keyboard to enter the invertelevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manholeelevation value manually*
Enter/<Select text of pipe size>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually*
For station 425.02 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *elect the invert elevation text on the drawing or enter Enter on the keyboard to enter the invertelevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manholeelevation value manually*
Enter/<Select text of pipe size>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually*
For station 649.73 ...
Enter/<Select text of Manhole ID>: *select the Manhole ID text on the drawing or enter Enter on the keyboard to enter the Manhole ID value manually*
Enter/<Select text of invert elevation>: *elect the invert elevation text on the drawing or enter Enter on the keyboard to enter the invertelevation value manually*
Enter/<Select text of manhole elevation>: *select the manhole elevation text on the drawing or enter Enter on the keyboard to enter the manholeelevation value manually*
Enter/<Select text of pipe size>: *select the pipe size text on the drawing or enter Enter on the keyboard to enter the pipe size value manually*
Another Polyline [<Yes>/No]? *enter Yes to input another trench reach from a polyline or enter No to finish*

At the end of the command, a file opening dialog would be prompted to you to specify a .sew file name to store the trench network structure.
Prerequisite: A drawing with one or more polylines that represent the trench structure.

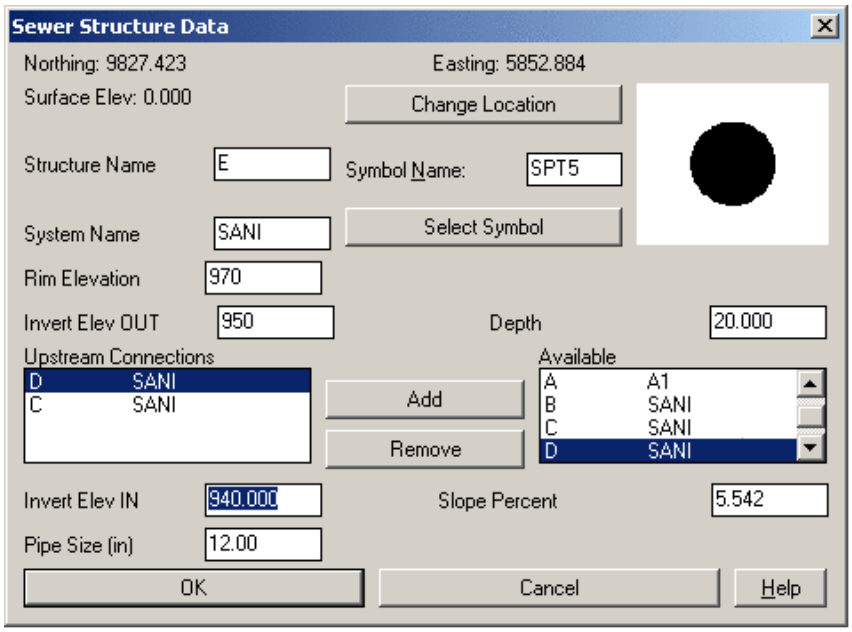
Keyboard Command: pline_trench

Create Trench Network Structure

Function

This command allows you to create or modify a trench network structure on a drawing. Before you are able to locate the trench structure, the drawing has to be open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. You can locate the trench structure by one of three methods: picking points on the drawing, entering the point number, or specifying the station and offset of a centerline. If you use centerline method, you need to specify a centerline file. After you locate a point on the drawing, you are prompted the **Sewer Structure Data Dialog** for entering the sewer structure

information, such as Structure Name, System Name, Symbol Name, and Elevations. Take a look at the list of the trench points that have been defined. If there is any point that is connected upstreamly to the current point, you add it to the Upstream Connections list. The Invert Elevation and the Pipe Size fields will be filled with the information of the upstream point. Click OK to finish entering the trench structure data. The command will repeatedly ask you to pick a structure point until you hit **Enter** to finish. The trench network structure data is saved in a .sew file .



Prompts:

By Pick:

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? *press Enter to do Pick point*
Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Pick structure location: *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pickstructure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *pick a point*
Sewer Structure Data Dialog: *enter trench structure information*
Pick structure location (Enter to end): *press Enter to finish*

By station-offset of CL:

Locate by pick point, point number or station-offset [<Pick>/Number/CL]? CL (enter CL to do locating

trench structure by station-offset of a centerline)
Specify a centerline file.

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Structure Station: 0 *(enter the station number on the centerline)*
Structure Offset: 200 *(enter the offset from the centerline)*
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): 100 *(enter the station number on the centerline)*
Structure Offset: 200 *(enter the offset from the centerline)*
Sewer Structure Data Dialog: *enter trench structure information*
Structure Station (Enter to end): *press Enter to finish*
Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface.

Keyboard Command: locate_trench

Edit Trench Network Structure

Function

This command edits the existing trench structure data on the drawing. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing. If there is no such structure in the .sew file, you would get a error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command will restore the trench structure data from the corresponding .sew file and display it on the Sewer Structure Data Dialog for editing. Click OK to confirm your modification. You are prompted to edit another structure point until you press Enter to finish. All modifications are saved in the .sew file.

Prompts:

Loading edges...
Loaded 4 points and 5 edges
Created 2 triangles

Select sewer structure to edit: pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information
Select sewer structure to edit (Enter to end): pick a point
Sewer Structure Data Dialog: modify its information

Select sewer structure to edit (Enter to end): press Enter to finish
Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: edit_trench

Remove Trench Network Structure

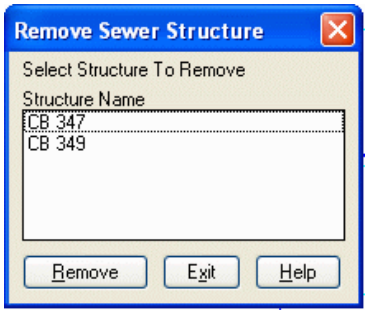
Function

This command removes the existing trench structure data. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name. The command first prompts you to pick a sewer structure on the drawing or to select from a List of your Sewer Structures. If there is no such structure in the file, you will get a error message like this: "Error: unable to locate structure in file C:\temp\takeoff\SANI1x.sew, otherwise this command removes the structure from both the drawing and and the .sew file immediately. You are prompted to remove another structure point until you press **Enter** to finish. The removed trench structure points would no longer be found in the .sew file.

Prompts:

Command: remove_trench

Select structures to erase by screen pick or name list [<Pick>/List]? Pick to choose from the screen, or List to choose from the below dialog.



Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

Keyboard Command: remove_trench

Find Trench Network Structure

Function

This command will center the screen and draw an arrow to the structure you specify.

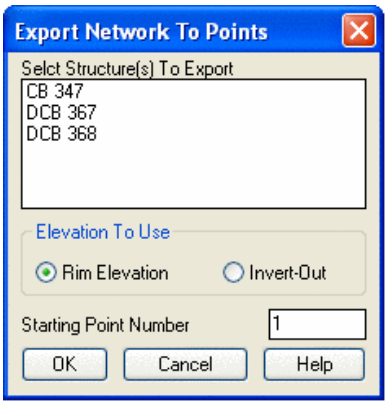
Prerequisite: a Trench Network

Keyboard Command: findswr

Export Trench Network Data

Function

Export to Points



This command will add points at your trench structures and add them into your coordinate file by either the Rim Elevation or the Invert-Out.

Export to Profiles



This command will create a profile file (.pro) of your trench either going Upstream or Downstream. The (.pro) file can then be drawn under Roads->Draw Profile.

Prerequisite: a Trench Network

Keyboard Command: swr2pts, swr2pro

Trench Network File Backup

Function

Save Trench Network File saves your trench network as a (.sew) file.

Load Trench Network File loads a previously saved (.sew) file.

Prerequisite: none

Keyboard Command: save_trench, load_trench

Draw Trench Network Plan View

Function

This command draws a trench network structure on the screen, based on the Plain Vew Label Setting command and the trench network structure data in the .sew file whose name is as same as the drawing name. If Takeoff couldn't find such file in the same directory where the drawing locates, nothing would be drawn on the screen.

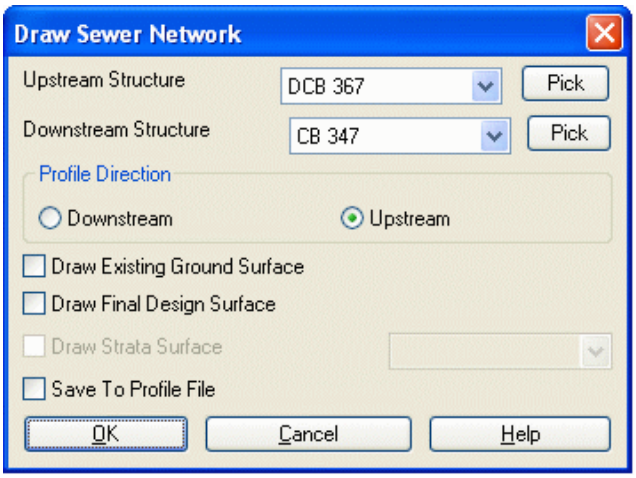
Prerequisite: A open drawing.

Keyboard Command: plan_trenc

Draw Trench Network Profile

Function

This command allows you to draw a branch of the trench network structure as a sewer/pipe profile. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file". The command first prompts you the **Draw Sewer Network Dialog**. Select the Upstream and Downstream Struct that you want to draw. If you want to draw the existing and final design surface, as well as Strata Surfaces, toggle on Draw Existing Ground Surface, Draw Final Design Surface, and Draw Strata Surfaces options. If your profile is from upstream to downstream, then select the Profile Direction as Downstream, otherwise Upstream. You can also choose to save the profile data to a profile file. Click OK to draw.



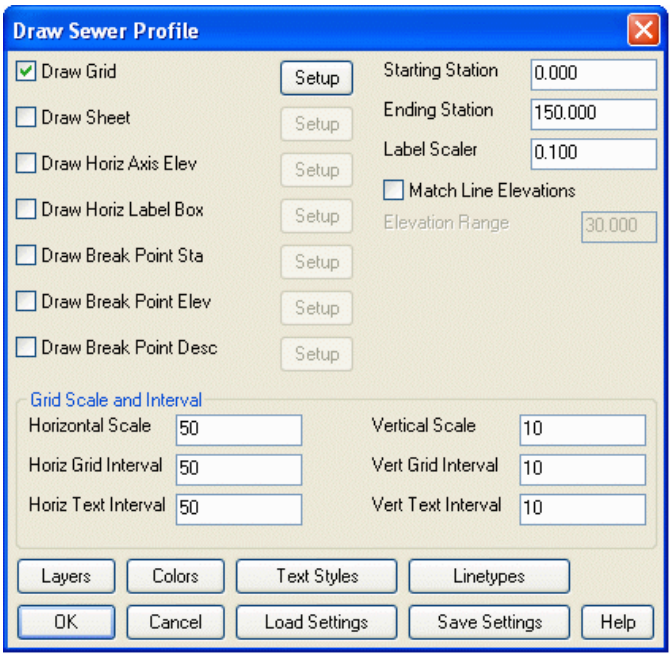
Prompts:

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles

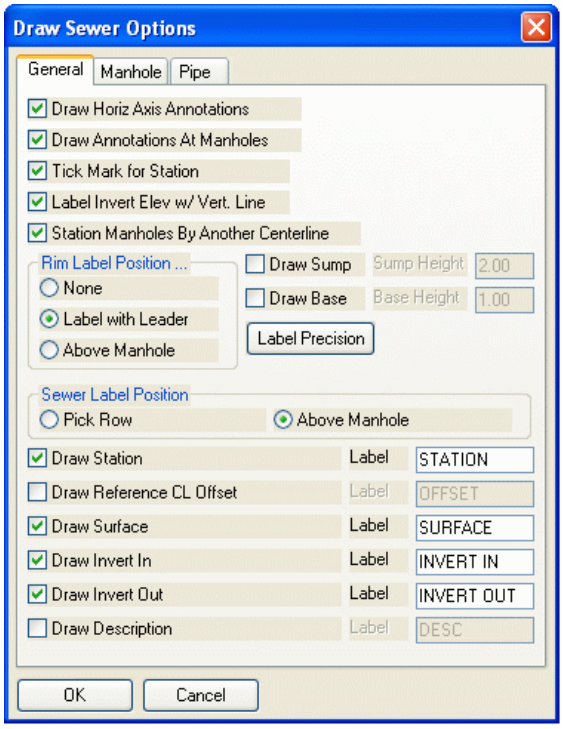
Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles

Initializing Draw Profile command ...
Draw Sewer Profile Dialog

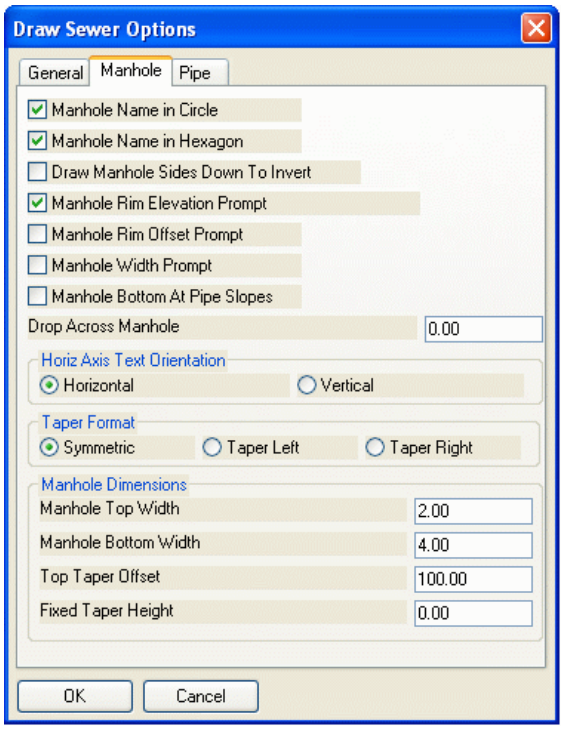
Enter drawing parameters such as Grid scale, text scaler, starting and ending stations etc. for drawing the sewer profile.



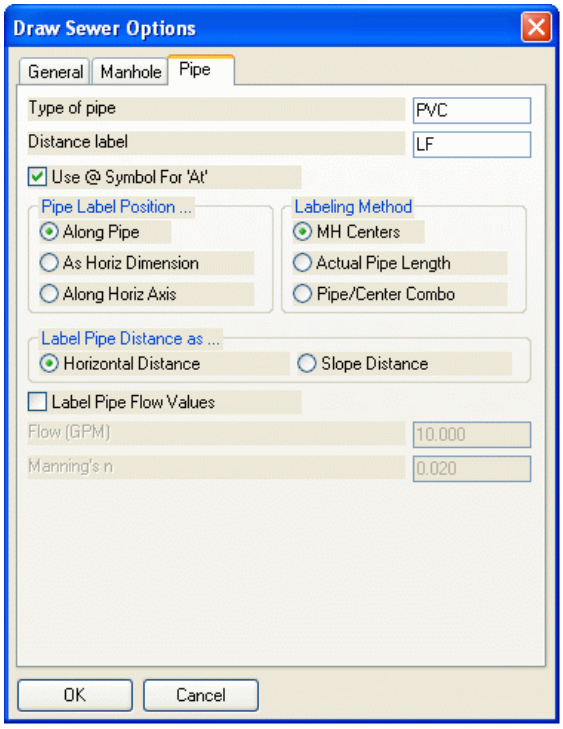
Enter general sewer profile settings such as elevations (Rim, Invert-In, Invert-Out) to draw and label.



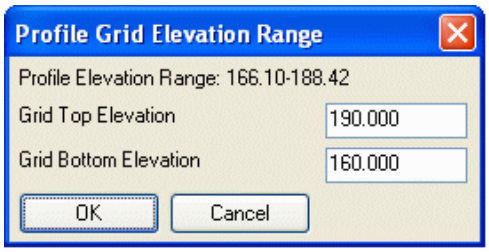
Use the Manhole tab to define what manhole information is labeled in your trench profile.



Use the Pipe tab to define what piping information is labeled in your trench profile.



The command will find the elevation range of your profile and display it at the top of this dialog. Here you can set the elevation top and botton of the profile's grid.

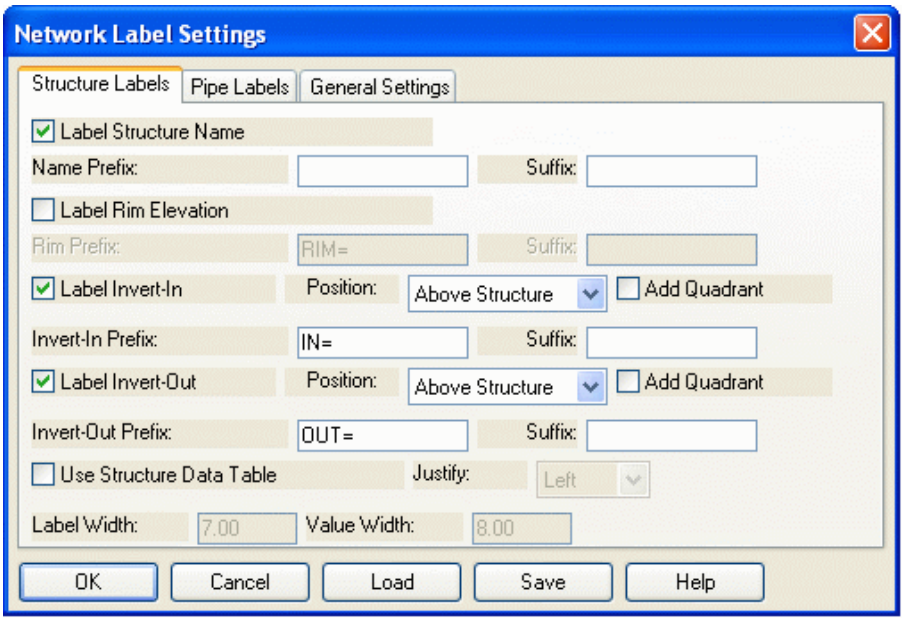


Prerequisite: Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.
Keyboard Command: profile_trench

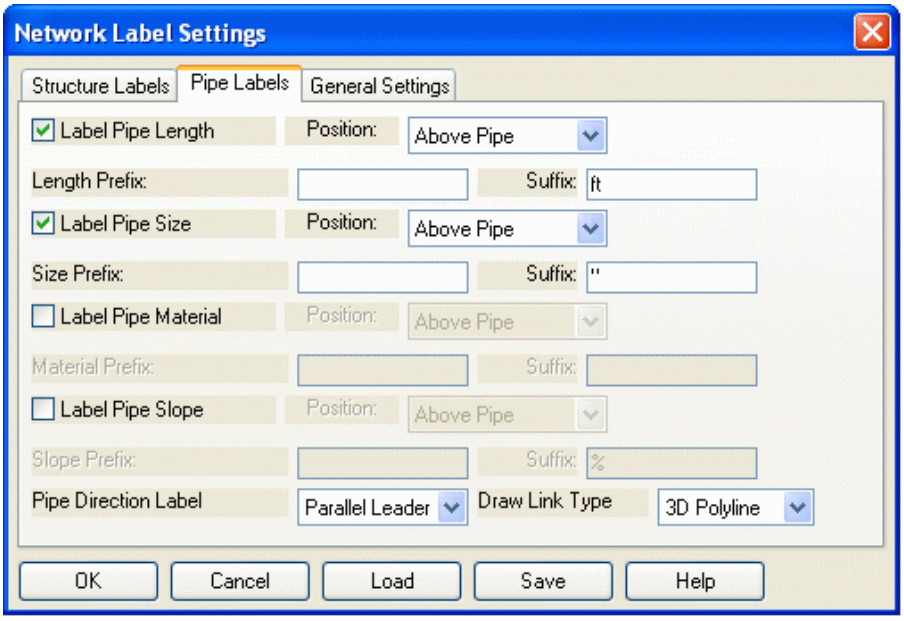
Plain View Label Settings

Function

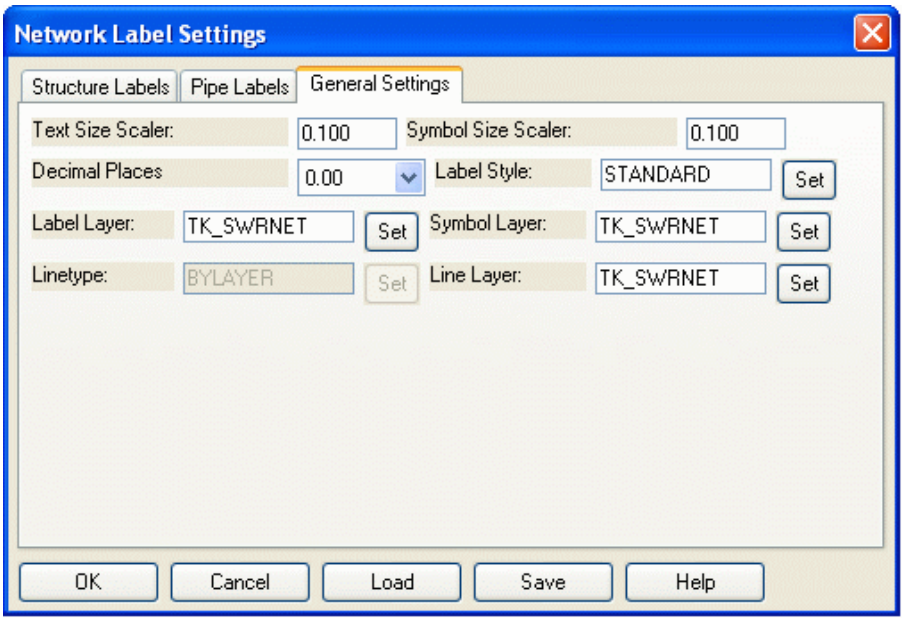
This command allows you to set the labeling for your structures and piping. The below dialog box gives you the option to display the Structure Name, the Rim Elevation, the Invert-In, and Invert-Out. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. The Use Structure Data Table will create linework around each Structure's labeling.



This below dialog box gives you the option to display the Length, Size, Material, and Slope for you Piping. In addition, you can set the Prefixes, Suffixes and labeling location as you so desire. To specify to which structure the label is intented for, select Arrow On Pipe, Parallel Leader, or None. You can also set the type of linework to draw.



In this dialog you can set the Porperties for your Symbol and Linework as well as the decimal places to report.



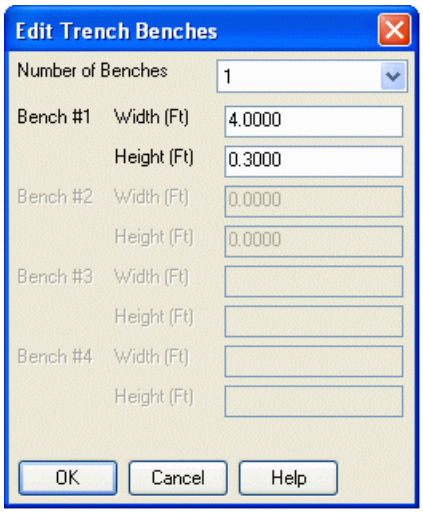
Prerequisite: a trench network

Keyboard Command: swrsetup

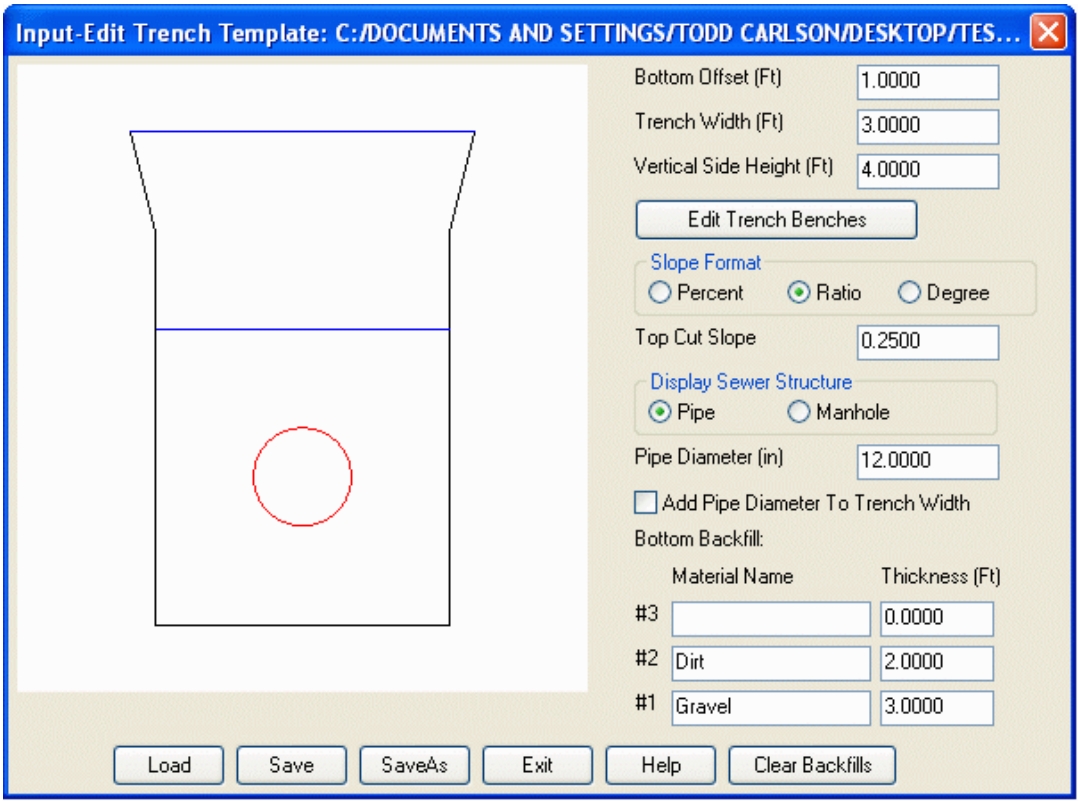
Input Edit Trench Template

Function

This command lets you create a new trench template or modify an existing trench template. It prompts you the **Input-Edit Trench Template Dialog**. If you are modifying a trench template, click the **Load** button on the dialog to open a trench template file and display the template data on thedialog. Enter the dimensions of the trench: bottom offset, trench width and vertical side height. The Edit Trench Benches button will bring up the below dialog, and allows you to enter in up to four benches into your trench.



There are three methods for entering the cut slope, Percent, Ratio and Degree. Choose one of the methods and enter the slope value. Display Sewer Structure allows you to see your pipe or manhole as part of the trench. This is for display purposes only, calculations will be drawn from the pipe size you set in the Trench Network Structure commands. Add Pipe Diameter To Trench Width will increase the size of your trench by the diameter of your different pipe sizes. There are three trench bottom backfill layers that can be defined. Enter the layer label in the material name field, the depth of the layer in the thickness field. Click Save or SaveAs to save the template information in a .tch file, and Click Exit to quit this command.



Prompts:

Input-Edit Trench Template Dialog
Enter the dimensions of the trench template, save the information to a template file (.tch).

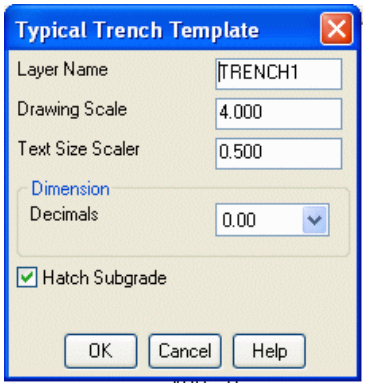
Prerequisite: None.

Keyboard Command: make_trench_tpl

Draw Typical Trench Template

Function

This command draws a trench template on the screen. After you select a trench template file (.tch) to draw, a **Typical Trench Template Dialog** is prompted for entering the layer name, drawing scale, text sizescaler and selecting how many decimal points you want. You can also hatch the backfill on the drawing. Click OK to draw the template at the position that you pick on the screen.



Prompts:

Pick position to draw template: *pick a position on the screen*
Prerequisite: None.

Keyboard Command: draw_trench_tpl

Trench Subgrade Areas

Function

The purposes of Trench Subgrades is to assign a different type of trench template when a trench passes under a road, building pad, etc.

Set Trench Subgrade Polyines

Choose a closed polyline that defines the area that you want a different trench template for, ie a building pad polyline.

Prerequisite: None.
Keyboard Command: tag_trench_subgrade

Clear Trench Subgrade Polyines

This command untags selected polylines for trench subgrade use.

Prerequisite: Trench Subgrade Polylines
Keyboard Command: untag_trench_subgrade

Hatch Trench Subgrade Area

This command hatches trench subgrades for easy viewing.

Prerequisite: Trench Subgrade Polylines
Keyboard Command: hatch_trench_subgrade

Erase Trench Subgrade Hatch

This removes previous made trench hatching.

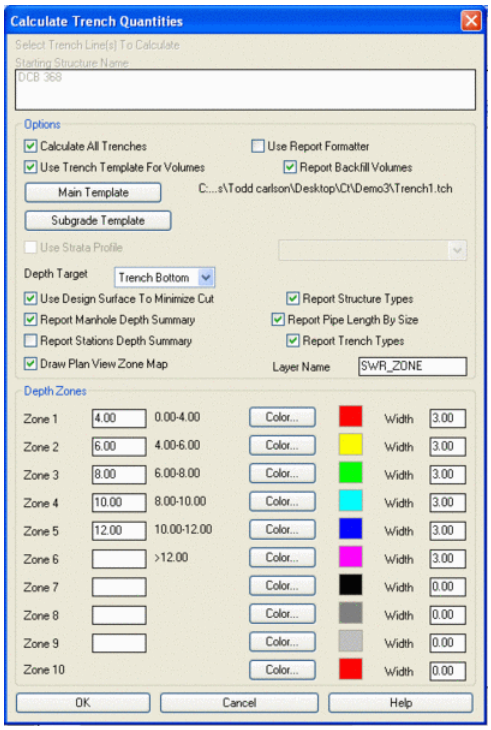
Prerequisite: Trench Subgrade Hatching
Keyboard Command: erase_trench_subgrade

Trench Network Quantities

Function

This command calculates the trench volumes. There has to be a trench network structure that has been created beforehand and its data is store in a .sew file whose name is as same as the drawing name, otherwise you would get a error message like "Error: no data in sewer network file".
The command loads the trench network data and split them into individual trench lines and display them on the **Calculate Trench Quantities Dialog**. You can choose to calculate the trench volume of one trenchline or several trench lines at a time. If you select **Use Trench Template for Volumes**, you need to **Set Trench Main Template** and **Subgrade Template** if you have one. If you select Use Design Surface Profile to Minimize Cut, then the Depth

Zones are depths from the final design surface, otherwise the Depth Zones are depths from the existing ground surface. If you have Strata Surfaces defined then the program can calculate cut volumes for a strata you select. Report Stations Depth Summary will report you your Trench depth zones by stations along the trench network. Trench Depths can be reported by either the bottom of the trench or bottom of the pipe by using the Depth Target pull-down. You can also color the trench in the drawing by defined zones. Click OK to compute the template volumes. Backfill quantities take into account pipe size. A report would be shown after the calculation.

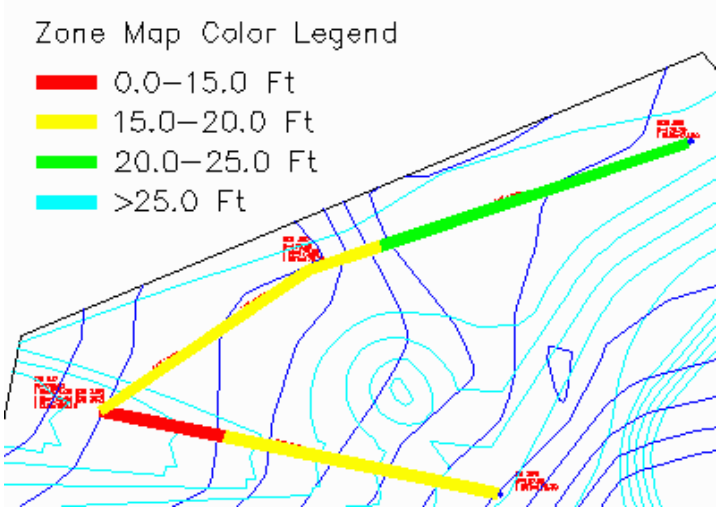


Prompts:

Loading edges...
Loaded 41041 points and 122901 edges
Created 81859 triangles
Loading edges...
Loaded 5487 points and 16332 edges
Created 10846 triangles
Trench Quantities Report Window

Draw zone map color legend on the screen [Yes/<No>]? *y for Yes*

Pick a point for color legend: *pick a point away from site*
Legend size <10.00>:*Press <Enter> for the default*



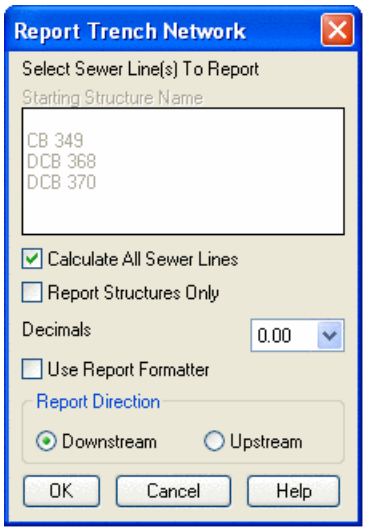
Prerequisite:
Your drawing is open, has been cleaned up and pre-processed by such commands as Define Layer Target, Set Boundary Polyline, Make Existing Ground Surface and Make Design Surface. Trench structure data has been stored in a .sew file, whose name is as same as the drawing name.

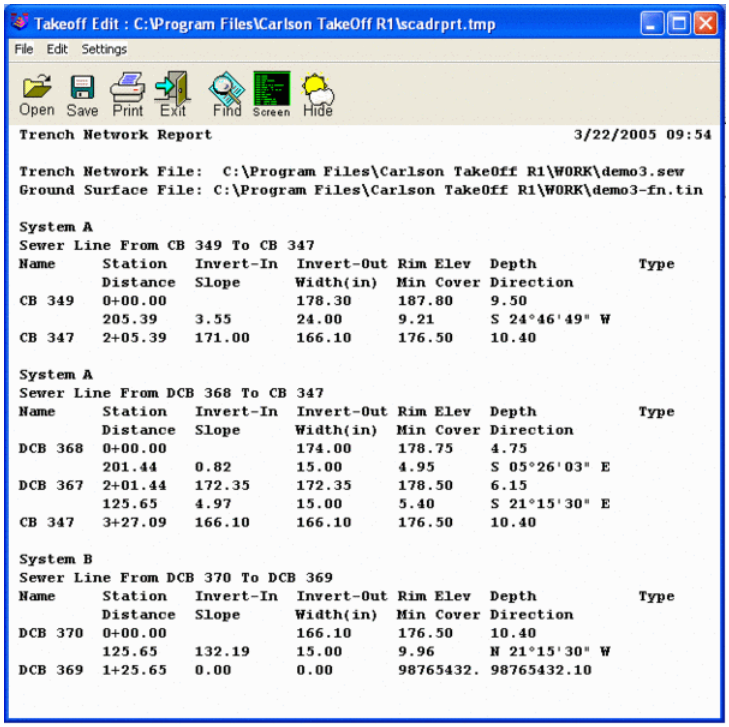
Keyboard Command: calc_trench

Report Trench Network

Function

This command will a report the Name, Station Distance, Invert-In Slope, Invert-Out Width, the Rim Elevation, Trench Type, Manhole Depth, and the Area Direction for the selected Trench. You may also choose to report the Trench Network from Downstream or Upstream, or just the Structures.




Takeoff Edit : C:\Program Files\Carlson TakeOff R1\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Trench Network Report 3/22/2005 09:54
Trench Network File: C:\Program Files\Carlson TakeOff R1\WORK\demo3.sew
Ground Surface File: C:\Program Files\Carlson TakeOff R1\WORK\demo3-fn.tin
System A
Sewer Line From CB 349 To CB 347
Name Station Invert-In Invert-Out Rim Elev Depth Type
Distance Slope Width(in) Min Cover Direction
CB 349 0+00.00 178.30 187.80 9.50
205.39 3.55 24.00 9.21 S 24°46'49" W
CB 347 2+05.39 171.00 166.10 176.50 10.40
System A
Sewer Line From DCB 368 To CB 347
Name Station Invert-In Invert-Out Rim Elev Depth Type
Distance Slope Width(in) Min Cover Direction
DCB 368 0+00.00 174.00 178.75 4.75
201.44 0.82 15.00 4.95 S 05°26'03" E
DCB 367 2+01.44 172.35 172.35 178.50 6.15
125.65 4.97 15.00 5.40 S 21°15'30" E
CB 347 3+27.09 166.10 166.10 176.50 10.40
System B
Sewer Line From DCB 370 To DCB 369
Name Station Invert-In Invert-Out Rim Elev Depth Type
Distance Slope Width(in) Min Cover Direction
DCB 370 0+00.00 166.10 176.50 10.40
125.65 132.19 15.00 9.96 N 21°15'30" W
DCB 369 1+25.65 0.00 0.00 98765432. 98765432.10

Prerequisite: a sewer line

Keyboard Command: reportswr

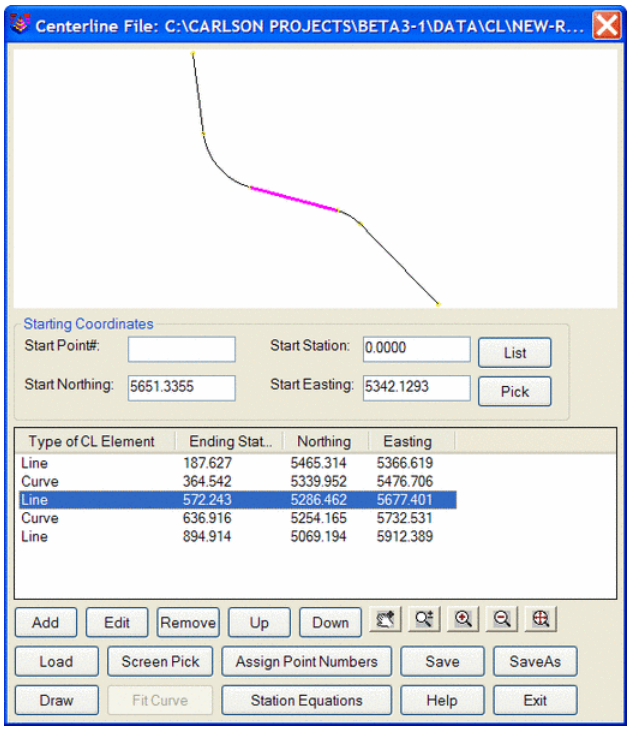
Roads Menu 14

Input-Edit Centerline File

Function

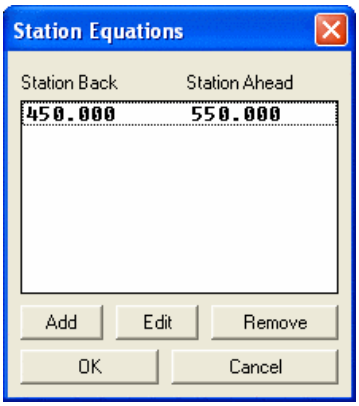
This command can be used to input a new centerline or edit an existing centerline (.CL) file. It can accept whatever information you have on your centerlines (coordinates, stationing, length of tangents and arcs, etc.). For creating a new centerline, it is ideal for entering data straight from highway design plans. For editing, this command allows you to change any of the geometric properties of any of the elements of the centerline (lines, curves, spiral-only and symmetrical spiral-curve-spiral elements), including the starting coordinates and station.

Starting this command launches the Centerline Input-Edit main dialog box. To edit an existing Centerline, you can either pick the Load button and pick the .CL file, or pick the Screen Pick button and pick the polyline in the drawing that represents the Centerline. The Centerline is then displayed in the graphics window of the dialog box. The highlighted segment in the text window is also highlighted in the graphics window.



Up/Down: Moves elements in the table Up and Down in the list. For example, if this centerline ended with a tangential line from the last curve, then was followed by a non-tangential line at 45 degrees NE, moving the last element up would create a line at 45 degrees after the curve (non-tangential), and the formerly tangential line will remain tangential and therefore continue at NE 45 degrees.
Draw: This button draws the centerline in the drawing on the specified layer.
Drag Action (Zoom and Pan): In the graphics window, hold the left mouse button down and move mouse to Pan, roll the wheel to Zoom.
Load: Loads an existing centerline (.CL) file for review or editing. After loading a centerline, the listbox in the dialog shows a list of all the elements in the centerline, identifying them as either a line, curve, spiral only or full spiral-curve-spiral element and reporting the ending station, northing and easting of the element.
Add: Adds a new element after the highlighted element. Prompts you for the type of the element to be added, Line, Curve, Spiral-Only or Spiral-Curve-Spiral.
Edit: Allows you to edit the highlighted segment.
Remove: Removes the highlighted element from the centerline.

Assign Point Numbers: This will create Carlson points along the elements of the centerline and store them to the current CRD file. The new points will be numbered in sequence beginning with the first available point number in the CRD file.
Save: Saves the currently loaded centerline to a file, or will prompt you for a name if no name has been set.
SaveAs: Prompts you for a file name for the saved file.
Exit: Exits this routine, prompting to save changes if necessary.
Help: Goes to the Help screen for the Input-Edit Centerline command.
Station Equations: At any number of locations on a centerline, you can set the back station and forward station for the re-stationing of the centerline. The station equation dialog appears below:



If the Station Back is lower than the Station Ahead, then a "gap" is inserted in the centerline, where the stations jump forward. If the Station Ahead is less than the Station Back, then an overlap occurs, where the common station range is repeated.

The dialog for every type of element shows the point ID, the northing, easting and station of the start point of the element. It then allows the user to modify or define the parameters specific to the type of element. The following are some of the things to remember about data entry in the centerline editor. These are valid for lines, curves and spirals.

- Wherever length of the element is to be entered, entering an expression of the type 123.5 - 93.7 would evaluate the difference of the values. This is particularly convenient where only the stations of the start and end points of the element are known.
- When the station is specified, the program takes the length of the element as the difference between the station of the start point of the element and the station specified.
- All bearings should be specified by entering the angle between 0 and 90 degree (in dd.mmss format) and selecting the quadrant.
- When entering the delta angle of a curve, only the absolute value (between 0 and 360 degree) is to be entered. The direction of the curve is to be explicitly set as right or left, the default being left. All angles are entered in (dd.mmss) format.
- Point numbers, when used, access their coordinates in the current .CRD file. If the point number specified has no coordinates stored in the coordinate file, the point number is remembered for that particular location (say the radius point of a curve or the SC point of a spiral). Then, when the .CL file is saved, the program creates points for that location and stores them to the .CRD file with the specified point number.

The dialog for a Line allows the user to specify the line primarily by its length or station and its bearing. The line can also be defined by its end point number or its coordinates. The bearing of a line can be changed if the Tangential to the Previous Element toggle is not checked. By default, any line which follows a curve element is defaulted to be tangential to it. To use a bearing different than that of the previous element, uncheck this toggle and enter the bearing.

The dialog for the Curve allows the user to define the curve primarily by its radius and delta angle or arc length. The other parameters of the curve that can be edited are the bearing of tangent-out and the "Station to", which also defines the arc length. The curve can also be specified by entering the coordinates or point numbers of its end point (PT) and the radius point. Another way to specify the curve would be to enter the chord length or PT point station and chord bearing. If the central PI point and a point on the forward tangent are known, then the curve can be defined by entering both of these points and at least one other property of the curve (like radius, arc length, delta angle). The point on the forward tangent can be any point that defines the tangent out direction including the next PI point. If only the central PI point is known, then the tangent-out can be entered by bearing instead of by forward tangent point. Central PI and forward tangent points are not displayed from the .CL file. They have to be entered by the user and are valid only for that particular edit session; that is, they are not remembered the next time the file is loaded. Curves are assumed to be tangent to the last element unless the Tangential to the Previous Element checkbox is cleared.

The Curve Edit Mode option defines how the curve is accepted in the centerline. If the Hold PC point is checked on, the radius is taken as fixed and the delta angle of the curve is calculated based on some additional parameter. Hence, the extent of the curve is unlimited. However, if the Hold PI points option is checked on, the bearing of tangent-out of the curve is taken as fixed and the radiusis calculated based on some other parameter. In this case, the curve is completely restricted within the central PI point and the bearing of tangent out. Hence, when the Hold PI points option is checked on, the above parameters should also be defined to carry out the calculations.

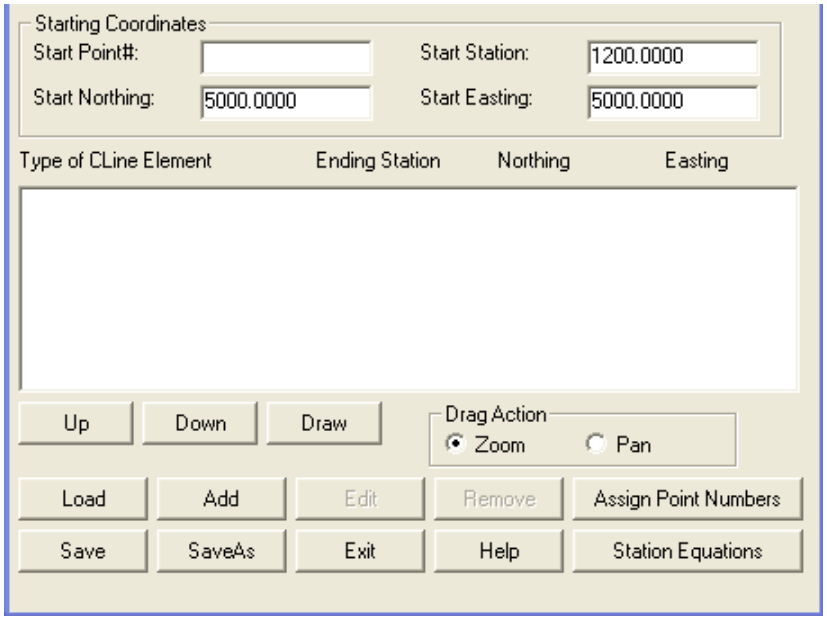
The dialog for the Spiral-Curve-Spiral element allows the user to define the spiral by entering either the various parameters of the spiral (like the angles and lengths) or the coordinates or point numbers of its defining points: the TS (Tangent-to-Spiral), SC (Spiral-to-Curve), Radius point, CS (Curve-to-Spiral), ST (Spiral-to-Tangent) and end point (optional). While defining the spiral by its geometric properties, the program will accept the data even if the information for the simple curve is given with zero spiral lengths. In this method, however, the central PI point of the spiral MUST be specified (that is, it is always in Hold PI Points mode). The tangent out can be defined by entering bearing or by specifying a point on the forward tangent. This forward tangent point can be the next PI coordinates. The direction of the spiral-in and spiral-out elements would be the same as the direction of the simple curve (left or right).

The spiral can be defined by several different parameters and the order that you enter data into the spiral dialog can be important. There are two main sequences for entering data. The method to use depends on the spiral data that you have. The first method is to enter the radius of the simple curve, the spiral in and out lengths, the tangent bearing out and the PI station. The second method is to make a Line segment coming up to the TS (tangent to spiral) point. This Line segment should be added before creating the Spiral element. Then with the Spiral In point set to the TS point, enter the radius of the simple curve, the spiral in and out lengths, the curve direction (left or right) and the arc length of the simple curve. Then the rest of the spiral points will be calculated.
The Spiral Only element allows for flexible transitions from curve to spiral to curve or line to spiral to curve or between any combination of curve and line elements. The Spiral-Curve-Spiral element, for example, can be entered as Line, Spiral Only, Curve, Spiral Only and Line, producing the same results.

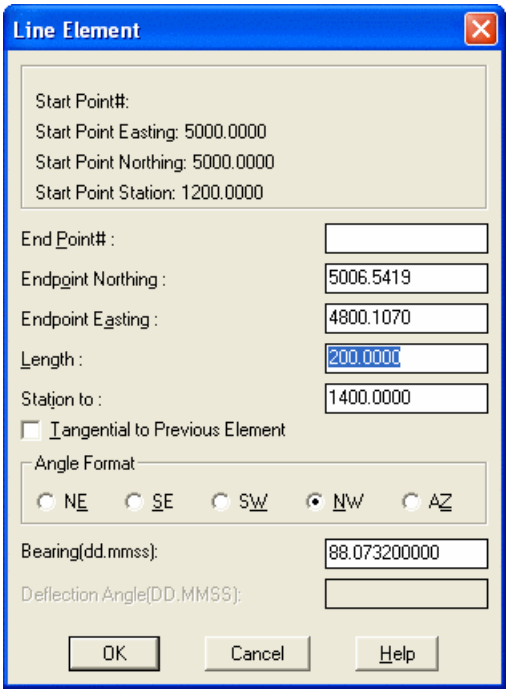
Once all the elements of the centerline are defined, the file can be saved and then plotted using the *Draw Centerline File* command.

Example
Here is an example of a highway interchange ramp that involves a starting tangent and a spiral curve that goes abruptly into a simple curve and then a final tangent. Here is the starting dialog.

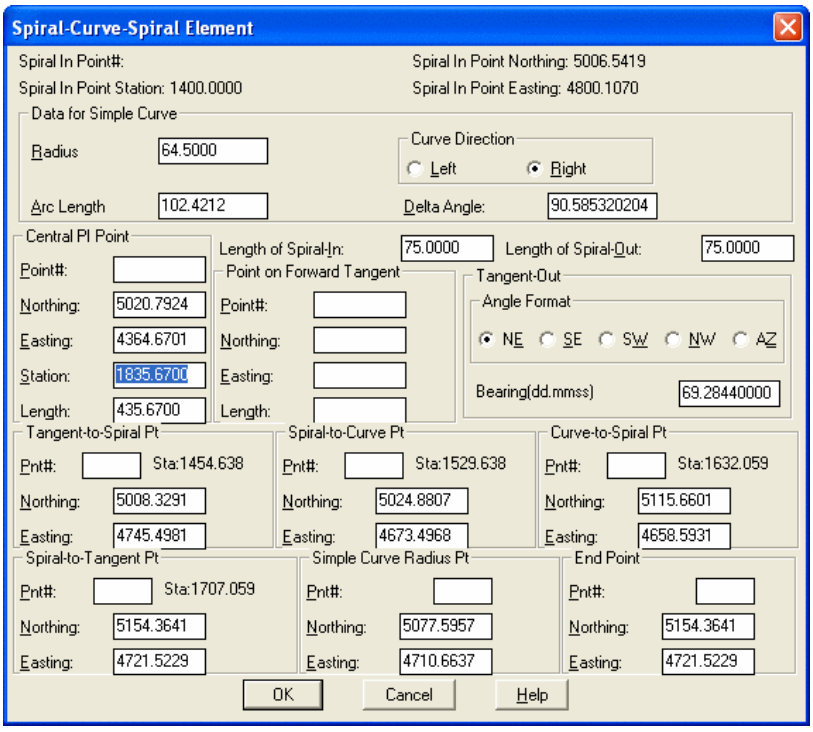
You start by entering a starting Northing and Easting and starting Station. The Start Point# is optional. Then the concept is that you click Add to add each subsequent element (line, curve, spiral-curve-spiral or spiral only).



Line (Tangent) Segment: We want to enter the tangent segment length up to the TS (tangent to spiral). Enter in the length (200.0), bearing (88.0732) and then the bearing quadrant (NW). Since the next spiral-curve-spiral element can be based on a PI station, it is not necessary for this line segment to go up to the TS point. The purpose of this line segment is to establish the tangent-in direction.

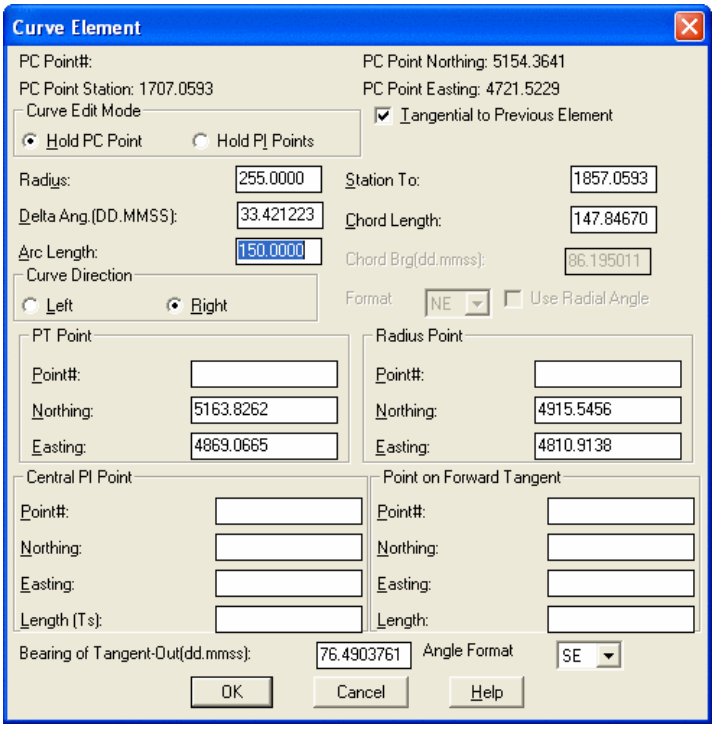


When OK is clicked, the routine will add the Line element as the first in the list of complete centerline elements. Next up is Curve-Spiral-Curve. Click Add.



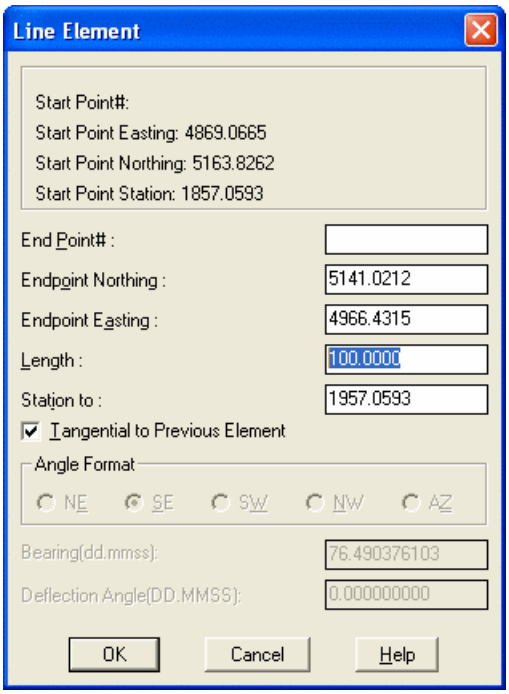
Spiral Segment: Though the dialog is complex (for total flexibility), the key on a typical symmetrical spiral curve is to enter four things: (1) the radius of the simple curve, (2) the spiral in and out lengths, (3) the tangent-out bearing and (4) the PI station (1835.67). Everything else will calculate when you press Enter for the PI station.

Curve Segment: Add the next element and select curve. The Curve dialog appears. The key is to enter the Radius Length (255), the Arc Length (150) and the Curve Direction. Everything else will calculate.

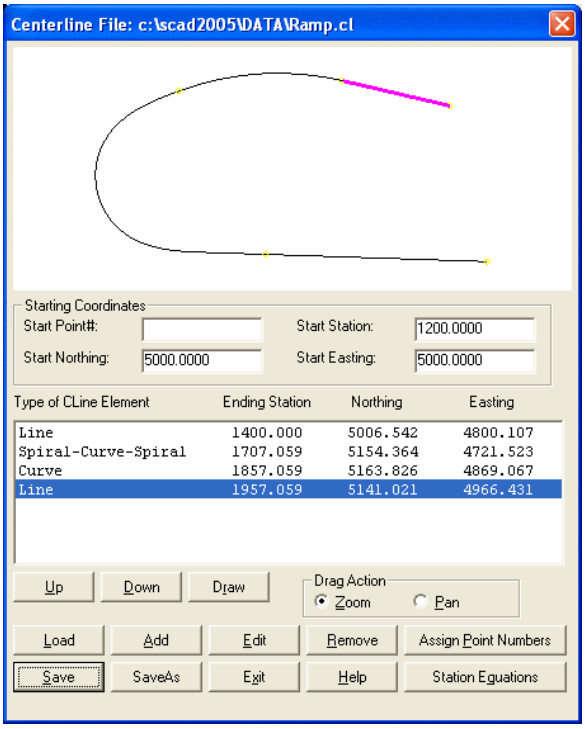


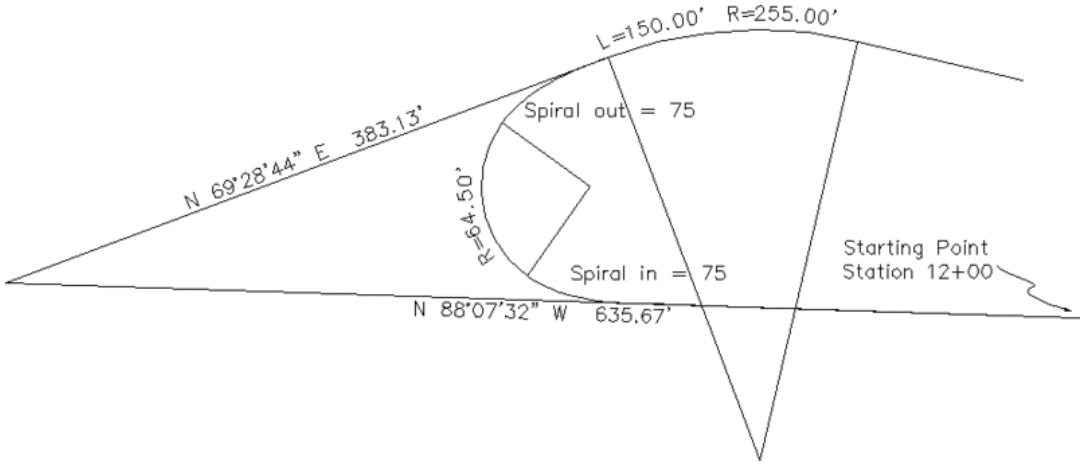
Final Line Segment: All you need to enter in the final dialog for the line (tangent) segment is its length. All other

items will calculate when you press Enter.



The completed centerline will appear as shown in the dialog and each element can be edited. Pick the Save button to store this centerline data to a .CL file.





Keyboard Command: cledit
Prerequisite: A CRD file to put points or take points from

Polyline to Centerline File

Function

This command writes a centerline (.CL) file from a polyline. The northing and easting for each vertex of the polyline is written to the centerline file and each arc in the polyline becomes a circular curve.

Prompts

Centerline file to Write

Enter the .CL file name to create.

Beginning station <0+00>: *Press Enter to accept 0+00, or type in the beginning station then press Enter***.**

Select polyline that represents centerline: *pick the polyline that represents your cemterline***.**

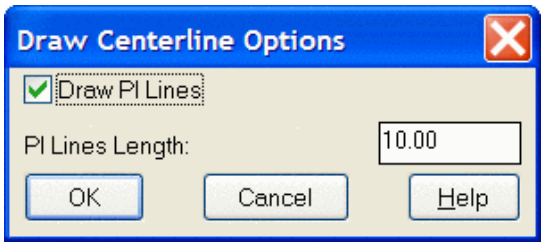
Prerequisite: a polyline

Keyboard Command: clpline

Draw Centerline File

Function

This command reads a centerline (.CL) file and plots it as a 2D polyline in the drawing at the proper coordinates. First you are prompted for the layer name for the polyline to be created. There is also an Options choice that allows you to specify whether to draw PI lines, and specify the length of same.



Next you are prompted for the file name of the centerline to plot.

The .CL file can be made with the following commands on the Roads menu:*Polyline to Centerline File and Input-Edit Centerline.* Drawing the centerline file is a way to check the .CL file data graphically for correctness. If a spiral exists in the .CL file, the spiral will be represented by polyline segments.

Prompts

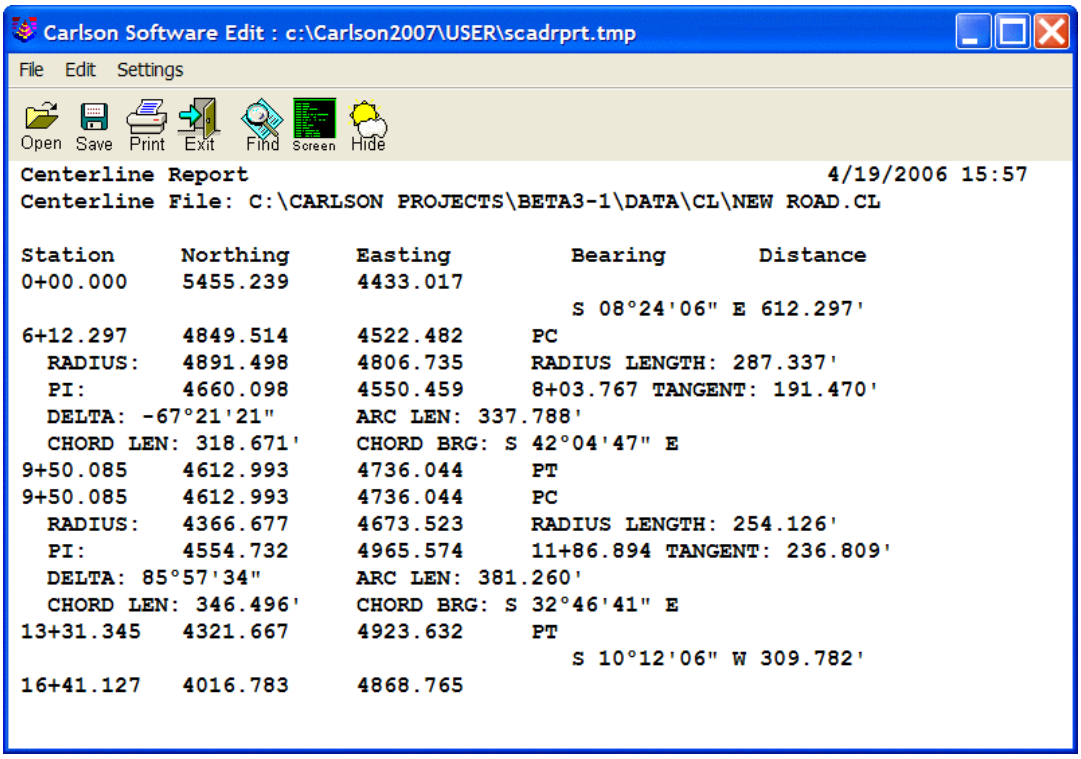
Options/Layer Name for Centerline <CLINE>: *press Enter* Enter the layer name to plot the polyline on.
Centerline File to Plot file selection dialog Select the .CL file name to read and plot.

Keyboard Command: cl2pline
Prerequisite: a centerline file

Centerline Report

Function

This command reads a centerline file and creates a report in the standard report viewer which can be written to a file, a printer, or to your drawing. If the centerline file contains point numbers, then the report will include these point numbers. If station equations are found, they are noted at the top of the report. Here is an example report:


Carlson Software Edit : c:\Carlson2007\USER\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Centerline Report 4/19/2006 15:57
Centerline File: C:\CARLSON PROJECTS\BETA3-1\DATA\CL\NEW ROAD.CL
Station Northing Easting Bearing Distance
0+00.000 5455.239 4433.017
S 08°24'06" E 612.297'
6+12.297 4849.514 4522.482 PC
RADIUS: 4891.498 4806.735 RADIUS LENGTH: 287.337'
PI: 4660.098 4550.459 8+03.767 TANGENT: 191.470'
DELTA: -67°21'21" ARC LEN: 337.788'
CHORD LEN: 318.671' CHORD BRG: S 42°04'47" E
9+50.085 4612.993 4736.044 PT
9+50.085 4612.993 4736.044 PC
RADIUS: 4366.677 4673.523 RADIUS LENGTH: 254.126'
PI: 4554.732 4965.574 11+86.894 TANGENT: 236.809'
DELTA: 85°57'34" ARC LEN: 381.260'
CHORD LEN: 346.496' CHORD BRG: S 32°46'41" E
13+31.345 4321.667 4923.632 PT
S 10°12'06" W 309.782'
16+41.127 4016.783 4868.765

Keyboard Command: clreport
Prerequisite: A centerline (.CL) file

Import Centerline

Function

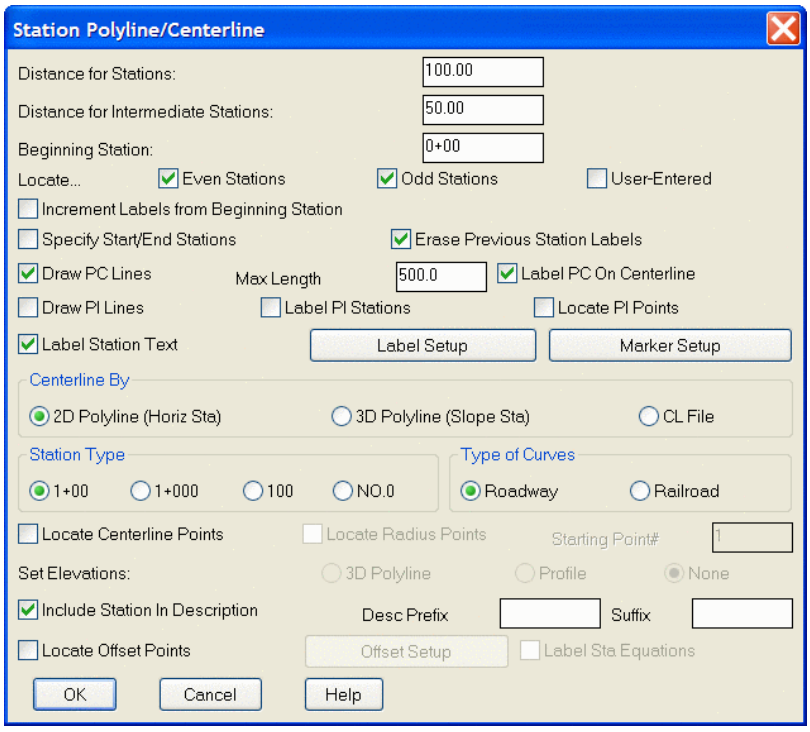
This command converts Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, and Leica road files into Carlson TakeOff centerline (.CL) files.

Prerequisite: a Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, or Leica road file

Station Polyline/Centerline

Function

This command will station a polyline or centerline file at a given interval distance. The options for this command are set in the dialog shown below. After setting the options, click OK on the dialog and then pick the polyline or select the centerline file.



Distance for Stations is the primary interval for stationing.

Distance for Intermediate Stations is the intermediate interval for stationing.

Beginning Station is the beginning station of the centerline for stationing.

Locate Even Stations labels the stations at the distance interval (i.e. 2+00, 3+00, etc.).

Locate Odd Stations labels the non-interval stations at the polyline/centerline end points and PC and PT points.

Locate User-Entered prompts you for individual stations to label.

Without the **Increment Station Labels from Beginning Station** option, the program increments the station labels from zero. For example, if the station interval is 100 and the polyline starting station is 145, then the program will label 2+00, 3+00, etc. With this option active, the station labels are incremented from the starting station. In this example, the program would then label 2+45, 3+45, etc.

When **Specify Start/End Stations** is checked, only the stations between and including the specified starting and ending stations will be labeled. If locate centerline points and offset points are toggled on, only points within the specified stations will be located.

When **Erase Previous Station Labels** is checked, previous station labels are erased when new ones are generated.

Draw PC Lines draws perpendicular lines at the starting and ending (PC and PT) stations of an arc of the polyline/centerline. The lines are drawn up to the radius point of the arc or to the distance set in the Max Length field.

Max Length controls the maximum length for the PC lines to be drawn described above.

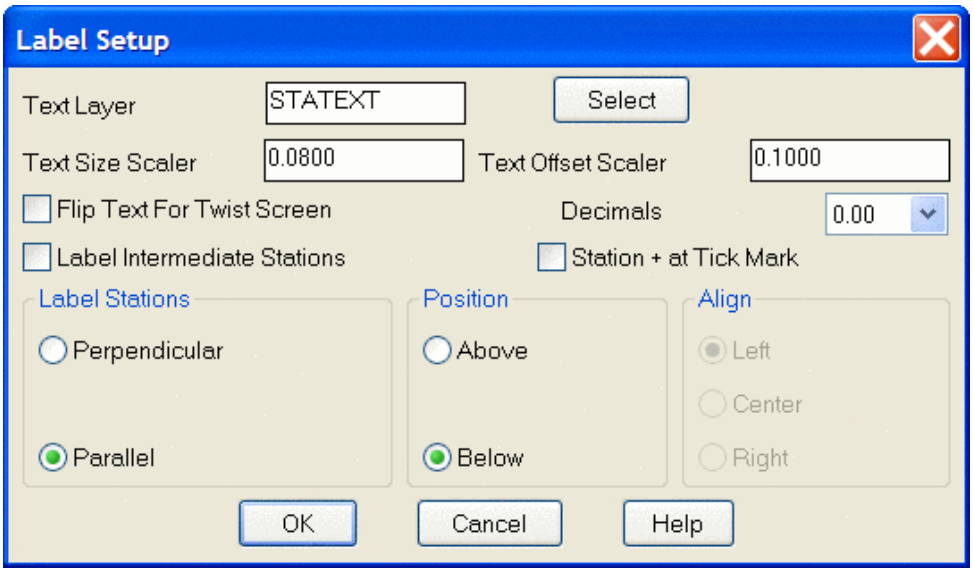
When **Label PC On Centerline** is checked, the station of the PC and PT will be labeled on the centerline as well as the PC and PT lines. When not checked only the PC and PT lines will be labeled.

Draw PI Lines draws a 2 segment polyline in both tangent directions from the PI as a marker for the PI.

When **Label PI Stations** is checked, the PI station is labeled at the PI point.

When **Locate PI Points** is checked a point will be created at the PI of a horizontal curve graphically and written to the active coordinate file.

When **Label Station Text** is checked, this command places station text along the polyline at the angle of the corresponding segment. After toggling this option on, the *Label Setup* button will become available for selection. Select it to configure the label settings as desired. Select the *Marker Setup* options to modify the size of the markers for certain types of stations. Seedefinitions following the dialog box.



Label Setup

- **Text Layer** is the user-specified layer for text labels to be drawn on.

- **Decimals** determines the number of decimal places of the stationing labels to be drawn for the odd stations and user entered stations only.

- **Text Size Scaler** determines the size of the station labels. This value multiplied by the horizontal scale setting in Drawing Setup results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scaler is set to 0.10, the station labels will be 10 units.
- **Text Offset Scaler** works like text size scaler above controlling the distance the text labels will be offset from the centerline.

- If the **Flip Text For Twist Screen** setting is checked and the drawing has been twisted using the twist screen command, the label text will be flipped to read in the proper direction of the stationing.

- **Label Intermediate Stations:** If the intermediate distance is the same as the station distance then no intermediate station ticks or labels will be drawn. For example, with the above entries and 0+00 for the first station the stations will be labeled with descriptions as follows: 0+00 0+50 1+00 1+50, etc.

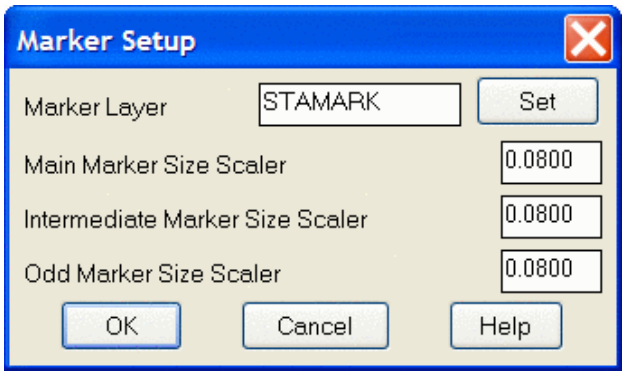
- **Station + at Tick Mark** labels the station text along the polyline with the '+' of the station text at the station's location on the polyline. See Marker Set up for marker size manipulation settings.

- Use **Label Stations** to specify whether to label the stations perpendicular or parallel to the centerline.

- Specify the **Position** of the station labels, either above or below the centerline. This is only available when labeling stations using the parallel option.

- **Align** determines the alignment of the station label, either left or centerline, centered along the centerline or to the right of the centerline. This option is only available when using the perpendicular option for station labels.

The **Marker Setup** options control the size of markers for different station types as well as the layer the markers will be drawn on.



Specify whether to define the **Centerline By** picking a 2D polyline or 3D polyline in the drawing or selecting a centerline (.CL) file.

- Using a **2D Polyline** will result in horizontal distance stationing along the polyline.

- Using a **3D Polyline** will result in the slope distance stationing along the polyline.

- Using a **CL File** will result in horizontal distance stations as with the 2D Polyline option only a prompt for the centerline to use will display.

Use **Station Type** to specify the stationing format to use.

Use **Type of Curves** to specify whether you are labeling a roadway curve (arc definition) or railroad curve (chord definition).

Locate Centerline Points will locate points and store them in the current CooRDinate file.

Locate Radius Points will locate the radius points of any arc segments.

Starting Point Number determines the starting point number for the points to be located.

There are two ways to **Set Elevations** for the centerline points and offset points to be created.

- The **3D Polyline** option gets the elevation of the point from a specified 3D Polyline within the drawing.

- The **Profile** option will determine the elevation of the point based upon the same station in the profile file. You will be prompted for the profile file to read for the elevation reference.

- With the **None** option selected, no elevations will be determined for the points.

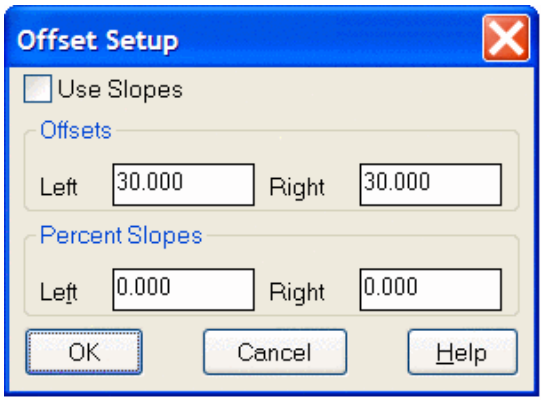
When **Include Station in Description** is checked, the station along the centerline will be included in the resulting offset point.

Description Prefix is an optional user-specified prefix to be added to the point description.

Description Suffix is an optional user-specified suffix to be added to the point description.

When **Label Sta Equations** is checked on any station equation, contained in a centerline (*.cl) file will be labeled. This option is only available when stationing a centerline file (*.cl).

Locate Offset Points will create points at the specified left and right offset distances from the centerline. Options for setting the elevations and descriptions of the points are available from the Offset Setup dialog.

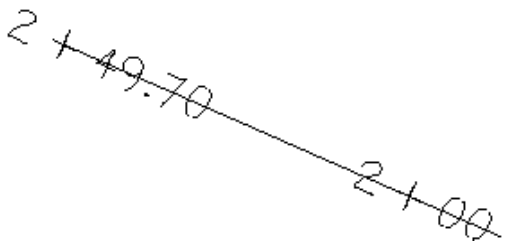


- When **Use Slopes** is on, it makes available the Percent Slopes fields for defining the slope from centerline both right and left for determining the elevations of the offset points.

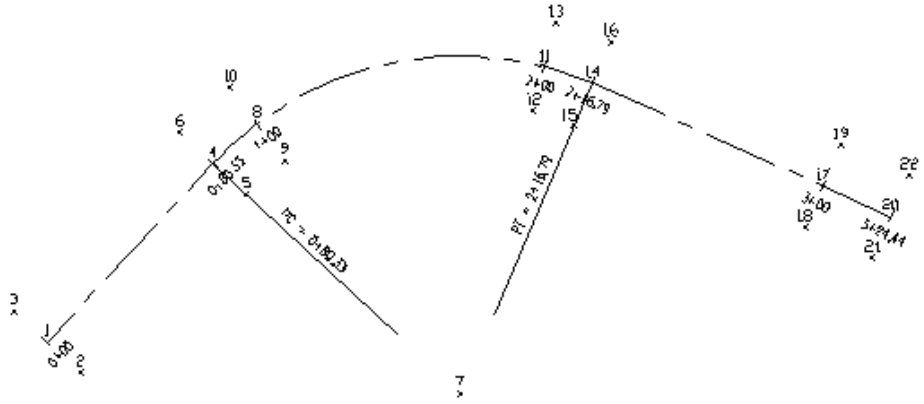
- Enter the desired **Offsets** left and right.
- Enter the desired **Percent Slopes** from centerline to the left and right offset points.

Prompts

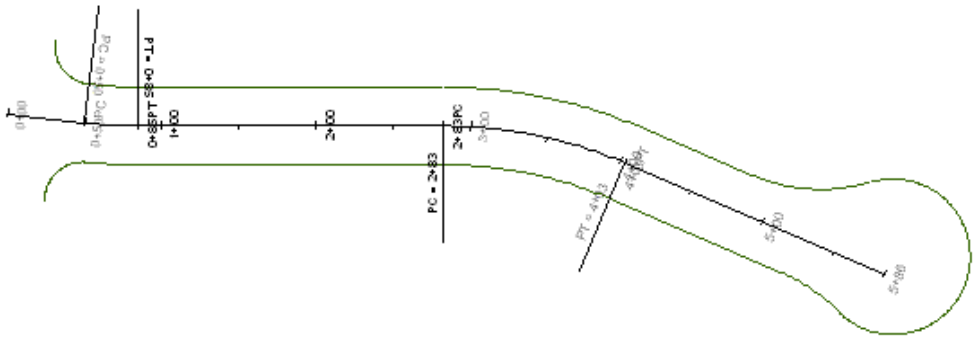
Station Polyline Dialog
Polyline should have been drawn in direction of increasing stations.
Select polyline that represents centerline: *select a polyline*



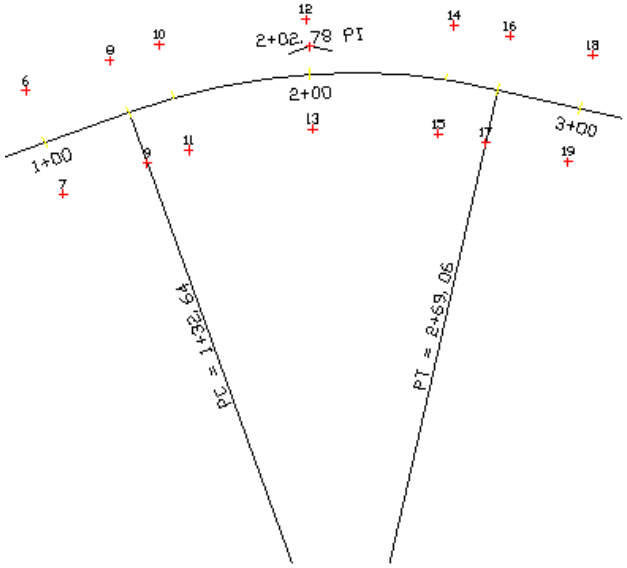
Closeup of Station + at Tick Mark option



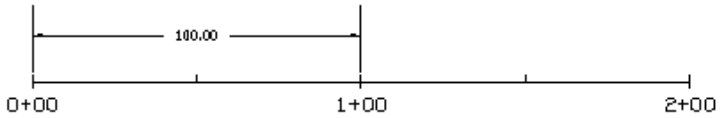
Labels with Label PC on Centerline checked on



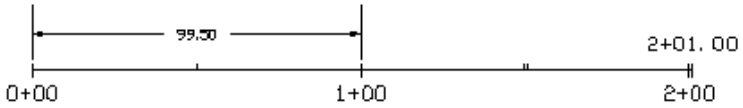
Labels set to perpendicular and Max Length of PC lines set to 75.0



Labels with Draw PI Lines, Label PI Stations and Locate PI Points all checked on



Labels using Centerline By 2D Polyline (Horzontal Station)



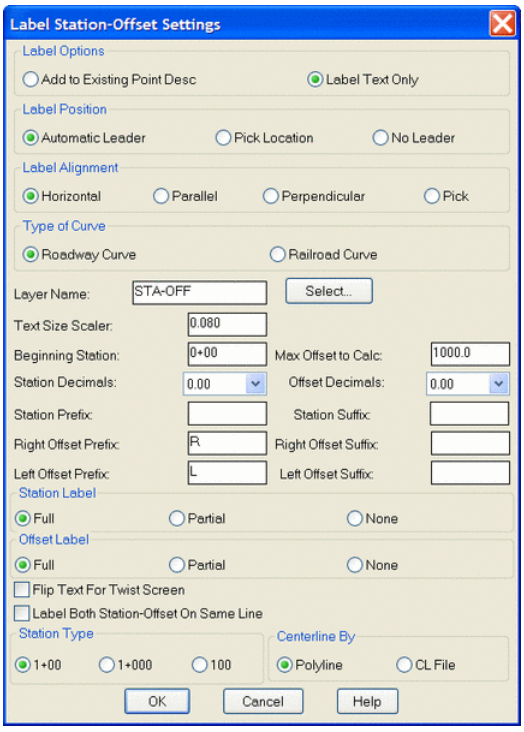
Labels using Centerline By 3D Polyline (Slope Station)

Keyboard Command: stapl
Prerequisite: A polyline or CL file

Label Station-Offset

Function

This command will label the station and offset of a point relative to a centerline. A polyline that represents the centerline or a centerline (.CL) file is required before using this command. The points to label can either be picked on screen or specified by point number. As the crosshairs are moved, the station and offset of the current position are displayed in real-time in a small window (see example). This command starts with the Label Station-Offset Settings dialog.



Label Options: Specify whether to label the text only or add the label to the existing point description.

Label Position: Specify if the program should automatically position the label with a leader, let the user pick the location, or use no leader.

Label Alignment: Specify whether the labels should be Horizontal on the screen, Parallel to the Centerline, Perpendicualr to the Centerline, or user-specified by Picking.

Type of Curve: Specify whether the centerline is for a roadway or railroad.

Layer Name: Specify the layer name for the labels.

Text Size Scaler: Determines the size of the labels. This value multiplied by the horizontal scale setting in *Drawing Setup* results in the size of the label. For example, if the horizontal scale is set to 100 and the text size scaler is set to 0.10, the labels will be 10 units.

Text Offset Scaler: Determines the text offset. This value works the same way as the Text Size Scaler.

Beginning Station: Specify the beginning station of the centerline. The polyline should be drawn in the order of increasing stations. Not available when you use a centerline (.CL) file to define the centerline.

Max Offset to Calc: Specify the maximum offset to calculate.

Station Decimals: Specify the display precision for the station text.

Offset Decimals: Specify the display precision for the offset text.

Station Prefix: Specify an optional prefix for the station text.

Station Suffix: Specify an optional suffix for the station text.

Offset Prefix/Suffix: Specify an optional prefix and/or suffix for each offset.

Station Label: Choose between Full label (1+35.42), Partial label (+35.42) or no station label.

Offset Label: Choose between Fill label (L15.35), partial label (15.35) or no offset label.

Flip Text for Twist Screen: With this option checked, the text will be flipped as necessary to accomodate the use of Twist Screen.

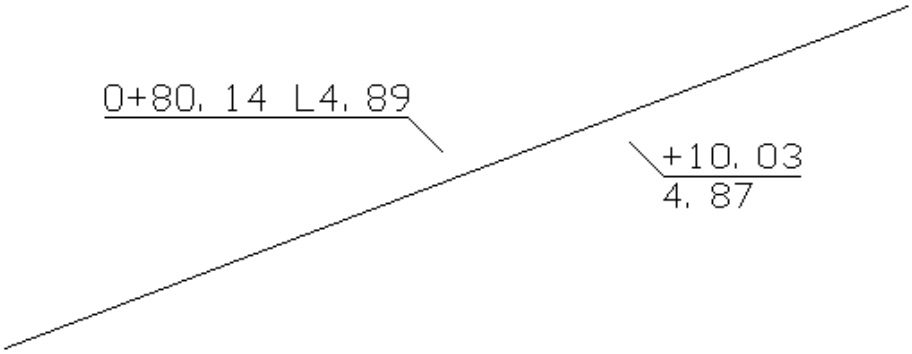
Label Both Station-Offset On Same Line: With this option checked, the station and offset label will be drawn on the same line.e e all

Station Type: Specify the stationing format to use.

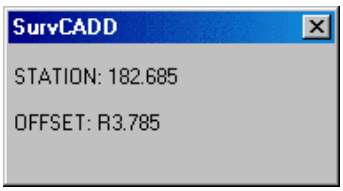
Centerline By: Specify whether to define the centerline by picking a polyline in the drawing or selecting a centerline (.CL) file.

Prompts

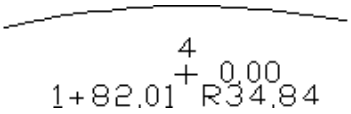
Label Station-Offset dialog
Polyline should have been drawn in direction of increasing stations.
Select Polyline Centerline: *pick the polyline centerline*
Pick point or point number (SS for Selection Set, G for Group, Enter to End): *pick a point*
Station on Line> + 2+10.91 Offset> 57.36 Right
Select point number to add station description to: *pick point number* This prompt will not appear if the L option, label only was selected.
Pick point or point number (Enter to End): *press Enter*



Top/Left Example: Label Text Only showing Full Labels and the Station/Offset on the same line
Bottom/Right Example: Partial Label



Real time display of station offset as you move the cursor



Add to Point Description format

Keyboard Command: offsta

Prerequisite: A polyline centerline

Offset Point Entry

Function

This command creates points along a centerline at specified stations and left and right offsets. The centerline can be defined by a polyline, centerline (.CL) file or two points. Offset Point Entry also has the ability to assign an elevation to the computed offset point based on a profile.

Store Points to CRD File: This option will store any points the the current coordinate (.CRD) file. This includes centerline points and offset points.

Locate Points on Centerline: When checked, the program will locate points along the centerline, otherwise just the offset points will be created.

Label Stations & Offsets: When checked, the program will label the station-offset as the point description attribute.

Use Reference 3D Polyline For Elevations: This option is to be used when creating points on centerline or storing the offset points to a coordinate (.CRD) file. When prompted to Select Elevation Reference Polyline, select the 3D polyline. Points will be created at the station entered and the offsets specified with the elevation of the same station along the 3D polyline. The option is good for creating points along the centerline for final grade elevation points. *Profile to 3Dpolyline* can be used to transfer the profile data to the polyline before calculating the final grade points.

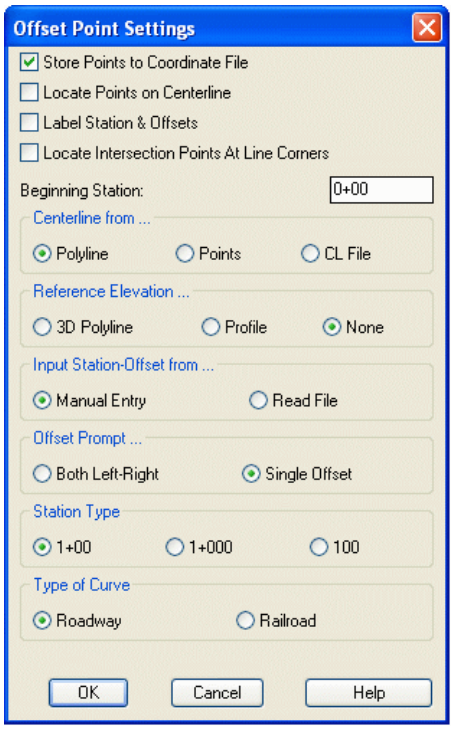
Centerline from: Specify whether to define the centerline by picking a polyline in the drawing, selecting a centerline (.CL) file, or using 2 points.

Input Station-Offset from: The Manual Entry option will prompt for the station and offset distances. The Read File option will read the stations and offsets from a text file. The text file format is comma delimited with point number, station, offset and elevation. The station should be just the station number without the '+' (ie. 250 instead of 2+50). The elevation is optional. The Read File option is a quick routine to convert a station-offset data file into coordinates.

Offset Prompt: When set to Both Left-Right, the program will prompt for left and right offsets. If you respond to an offset prompt with zero (0), no offset point is created. The Single Offset option will prompt for one offset per station. Enter a right offset with a positive value and a left offset as a negative value.

Station Type: Specify the stationing format to use.

Type of Curve: Specify whether the curves are for a roadway or railroad.



Prompts

The routine will prompt for a station and offset, as shown below:

Distance input/<Station along centerline ([Enter] to end)>:

However, if the "Profile" option is selected, prior to the first station prompting, and after selecting the centerline to process, the routine will prompt for a profile to process. The elevations are then computed automatically as each station and offset is entered. The profile option, when used in conjunction with "Store Points to∗.CRD File" will store the points with the interpolated elevations from the profile. Otherwise, the prompting is as follows:

Offset Point Settings Dialog

Polyline should have been drawn in direction of increasing stations.

Select Polyline near endpoint which defines first station.

[nea on] Select Polyline to Station-Measure: *(select polyline)*

(5309.0 4845.0) Station: 0.00

(5526.0 4917.0) Station: 228.63

Distance from beginning station along centerline (Enter to end): *110*

Starting Segment Station: 0.0 Ending Segment Station: 228.633

Working Line segment...(5413.4 4879.64 0.0)

Left offset distance <10.0>: *15*

Right offset distance <15.0>: *20*

Distance from beginning station along centerline (Enter to end): *[Enter]*

Pressing [Enter] with a blank response ends the command.

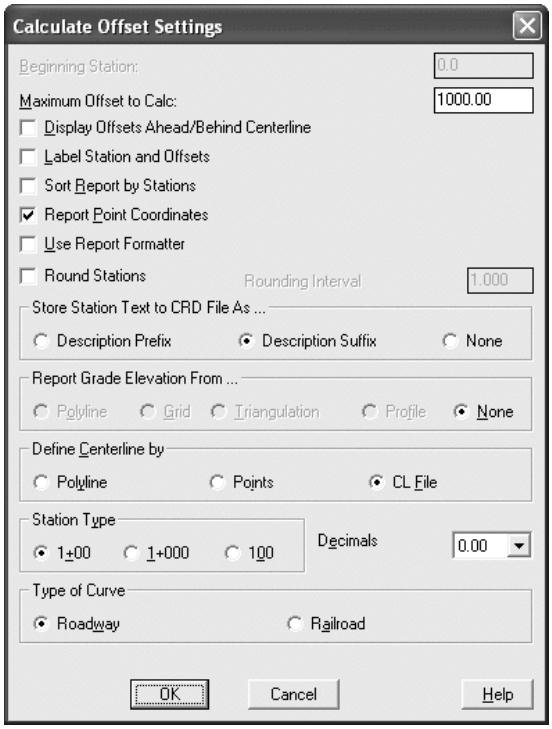
Keyboard Command: offpts

Prerequisite: a centerline (.CL) file, polyline or two points

Calculate Offsets

Function

The Calculate Offsets command computes and displays the station and offset of any picked point or point number, where the centerline can be a centerline file, a series of points or a polyline picked on the screen. When point numbers are entered, the computed station and offset can be added to the existing point description either as a suffix or prefix. The prefix/suffix options appear in the dialog shown below:

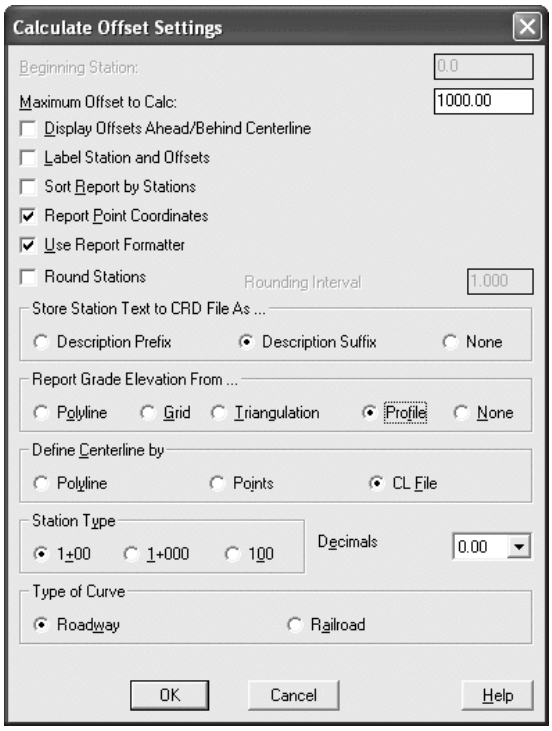


In the graphic shown here, the station and offset of points 8 and 9 are calculated using the "Description Prefix" and point 10 is calculated using the "Description Suffix".

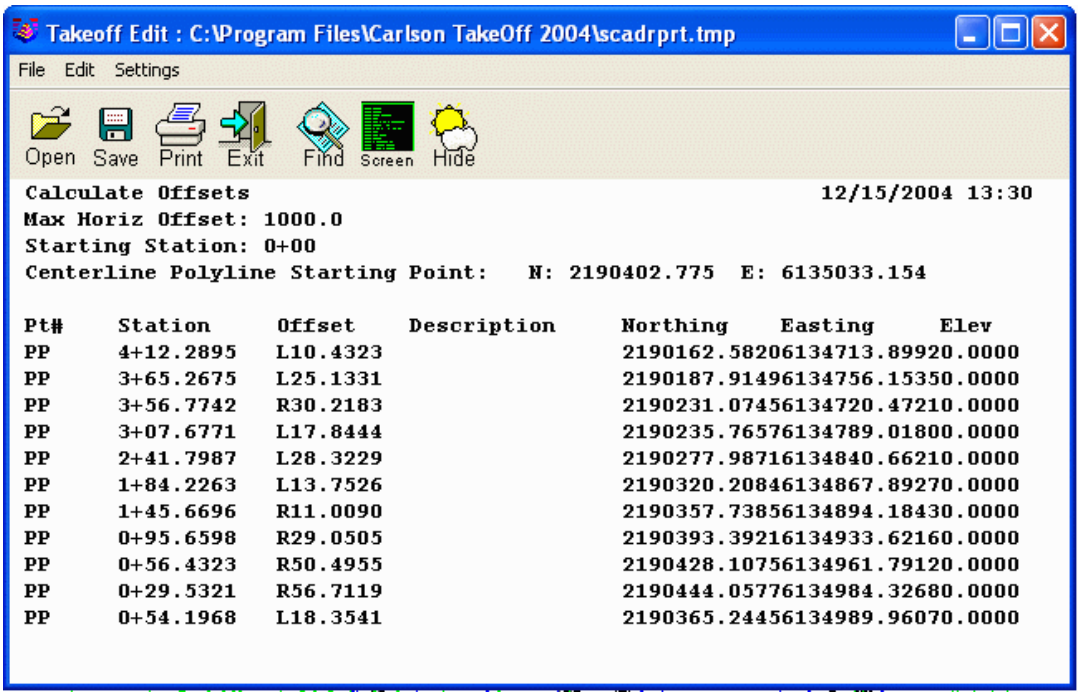


Reporting Elevations in Calculate Offsets

If the Report Formatter option is clicked on within Calculate Offsets, options appear to assign elevations to the picked or selected points, which can then be displayed. Special "Grade Elevations" can be computed from a 3D polyline, grid or triangulation file. Also, elevations associated with the station and offset can be calculated from a profile. The Grade Elevation is distinct from the known elevation associated with a point. The Grade Elevation, therefore, is only displayed within the Report Formatter option.



Here is a sample of a report:



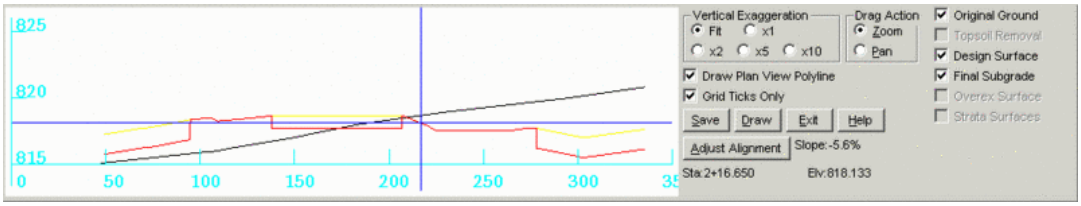
Prerequisite: a centerline file or a polyline to use as a centerline

Keyboard Command: calcoff

Quick Profile From Surfaces

Function

This command allows you to create, view, edit, and report profiles from the TakeOff surfaces.



Pick starting point (CL-Centerline,P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

The far right dialog box allows you to toggle on and off different Surfaces to view in the profile viewer including: Original Ground, Topsoil Removal, Design Surface, Final Subgrade, Overex Surface, Strata Surfaces. If a

surface is not defined in the current TakeOff project, like Topsoil Removal in this example, than you will not have the option to display it. In this example, the three Surfaces that can be displayed, Original Ground, Design Surface, and Final Subgrade, are displayed in the profile viewer.

When you move the cursor around the profile viewer a crosshair follows along the surface and reports the Station, Slope %, and Elevation at each point. It is displayed towards the bottom-right side of the screen next to Adjust Alignment. In this example the station is 2+16.650, the Slope is -5.6%, and the Elevation is 818.133. A crosshair can been seen in the profile drawing and along the alignment in the main drawing as well.

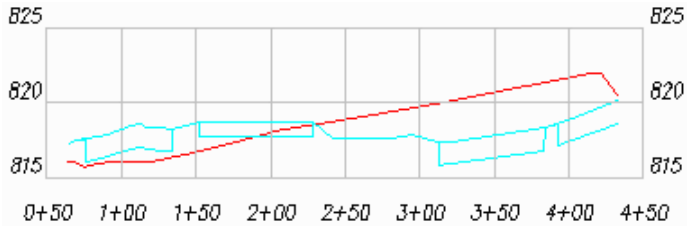
Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the elevation differentiation and different surfaces. The option Fit automatically exaggerates the vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To zoom in click and drag up, to zoom out click and drag down. To Pan, click amd drag the direction you want to move.

The Adjust Alignment icon allows to pick the polyline or centerline that you used and move it to your liking. If you selected an endpoint vertex, you can pivot that vertex around 360 degrees and the profiles will update in real time. This is helpful when checking for spikes. If you select the middle vertex then you can shift the entire centerline around.

If you created a profile alignment by picking points and you want to save that polyline you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View Polyline than the polyline will be lost when you exit out of the Quick Profile command. Grid Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical and horizontal scale as desired, pick a starting point to draw, and the profile is created. Note: the below example has a veritcal scale of 5 feet per grid and a horizontal scale of 50 feet per grid.



Prompts

Command:
QUICKPRO
Pick starting point (CL-Centerline,P-Polyline): p

Polyline should have been drawn in direction of increasing stations.

CL File/<Select polyline that represents centerline>:
Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Prerequisite: a surface

Keyboard Command: TK_QUICKPRO

Profile From Existing Surface

Function

This command will create a profile file (.pro) for the existing surface. To define the profile alignment, type in CL in the command prompt, and select a centerline file, or pick the polyline from the screen. This will create the profile file. You can now use the other Profile commands to draw, edit and report from this profile.

Prompts

Command: progrid2

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Enter the starting station <0.0>:

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Found 19 profile points.

Prerequisite: a surface

Keyboard Command: progrid2

Profile from Design Surface

Function

This command will create a profile file (.pro) for the design surface. To define the profile alignment, type in CL in the command prompt, and select a centerline file, or pick the polyline you want to use from the screen. This will create the profile file. You can now use the other Profile commands to draw, edit, and report from this profile.

Prompts

Command: progrid3

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Enter the starting station <0.0>:

Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Found 19 profile points.

Prerequisite: a surface

Keyboard Command: progrid3

Design Road Profile

Function

This command is for simultaneously creating a .pro file and drawing the road profile. The procedure is to first specify the on-screen grid and then enter or pick the stations and elevations.

Once two segments have been entered, you will be prompted for the vertical length. If you don't want a vertical curve, enter 0. Otherwise you can directly enter the vertical curve, or enter the sight distance or the K-value from which the vertical curve is calculated. The vertical curve can also be specified to pass through a point or do a best fit through multiple points. This through point option would be useful for hitting an existing feature such as a driveway on the vertical curve. Unequal vertical curves is another option where the vertical curve length going into the PVI differs from the length leaving the PVI. Before using your entry, the vertical curve, sight distance, and K-value are displayed. Object height and eye height are two variables that effect the vertical curve.

Notice that the station, elevation, and slope at the current position of your cursor crosshairs are displayed in real-time in a small dialog.

Prompts

File Selection dialog Specify a profile file to create.

Profile Settings dialog

Station of first PVI or pick a point: *0*

Elevation of PVI: *565*

Station of second PVI or pick a point ('U' to Undo): *200*

Percent grade entry/Ratio/<Elevation of PVI>: *575*

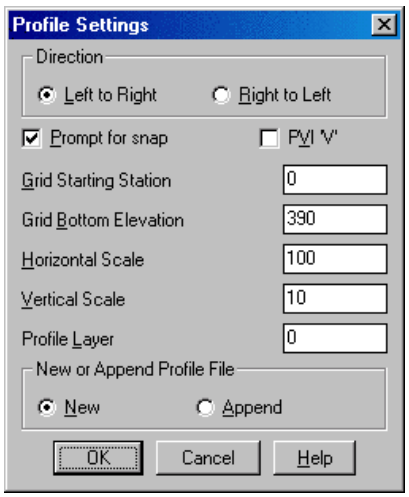
Station of next PVI or pick a point ('U' to Undo, Enter to End): *pick a point*

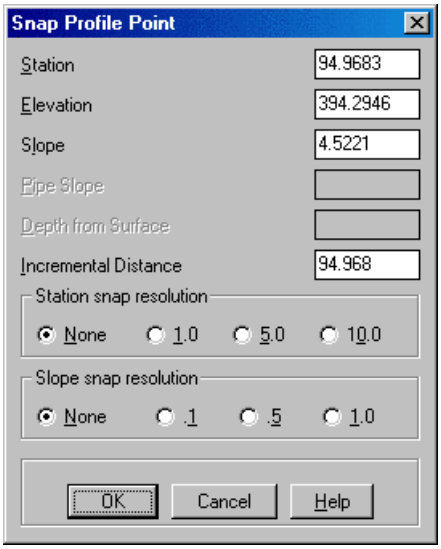
Snap PVI dialog

The Snap PVI dialog box appears when you pick a point (if the Prompt for Snap option in the Profile Settings dialog is selected). The station and slope may be changed to the nearest snap value. The elevation is the free variable and

it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.

View Table/Unequal/Through pt/Sight Distance/K-value/<Length of Vertical Curve>: *100*





For Crest with Sight Distance>VC and Vertical Curve => 100.00

Sight Distance => 87.30, K-value => 44.1

Use these values (<Y>/N)? *Press Enter*

Station of next PVI or pick a point ('U' to Undo, Enter to End): *press Enter*

Vertical Curve Text Options dialog box

Pick vertical position for VC text: *Pick a position above the profile grid.*

Prerequisite: A profile grid

Keyboard Command: road

Design Sewer/Pipe Profile

Function

This command creates a sewer profile (.PRO) file and draws it on the screen. It requires that a grid is already drawn. It begins with the Profile Sewer Settings dialog box.

Prompts

Bottom Manhole Width: Specify the size for the bottom of manholes. Not available when Profile Type is set to pipe.

Max Pipe Length: Specify the maximum limit for the distance between manholes.

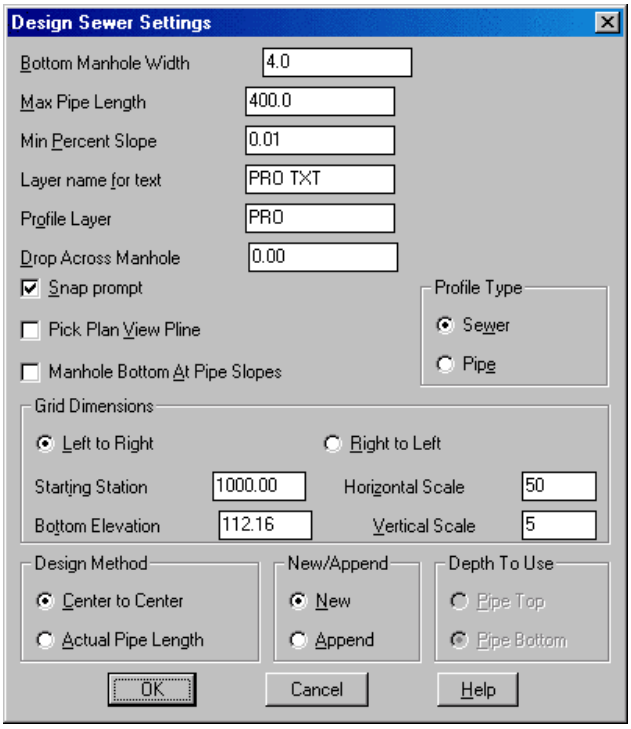
Min Percent Slope: Specify the minimum slope (absolute value) between manholes.

Layer name for text: Specify the layer name for annotation. If you enter a layer that does not exist, it will be created.

Profile Layer: Specify the layer name for pipes and manholes. If you enter a layer that does not exist, it will be created.

Drop Across Manhole: Specify the amount the elevation drop across the manhole in the direction of the profile. Will accept a negative a value. Not available when Profile Type is set to pipe.

Snap Prompt: Activates the PVI Snap dialog box. See below for description.



Pick Plan View Polyline: Allows you to select a polyline from plan view that represents the sewer center-line.

Manhole Bottom At Pipe Slopes: When checked, the manhole bottom will be drawn level with the pipe slope.

Profile Type: Choose between Sewer profile or Pipe profile. Pipe profile do not include manholes.

Grid Dimensions: Specify the grid dimensions on which the sewer will be designed.

Design Method: Choose whether distances specified are center or manhole to center of manhole or actual pipe length. Not available when Profile Type is set to pipe.

New/Append: Choose between creating a new profile (.PRO) file or appending an existing file.

Depth to Use: Choose between specifying pipe top or pipe bottom elevations. Not available when Profile Type is set to sewer.

File Selection dialog

Choose a new profile file name to create.

Pick Lower Left Grid Corner <5000.0,5000.0>[endp on]: *Pick the corner*

Select existing ground polyline or ENTER for none: *You may optionally pick a polyline to use for calculating the depth from the surface as the sewer stations are entered.*

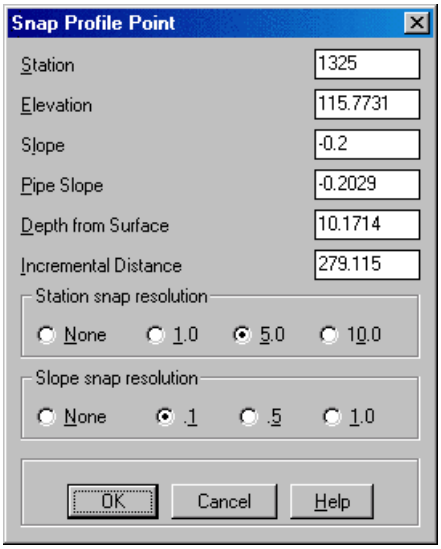
Station of first manhole or pick point: *0*

Invert Elevation of Manhole: *910*

Enter the step up/down in feet <0.00>: *Press Enter*

Station of second manhole or pick a point (U,E,D,Help): *pick a point*

If the Prompt for Snap option was selected in the main dialog, then the Snap Profile Point dialog appears here. The station and slope may be changed to the nearest snap value. The elevation is the free variable and it will change to compensate for any snap. To change the elevation, select the elevation edit box and enter the new value.

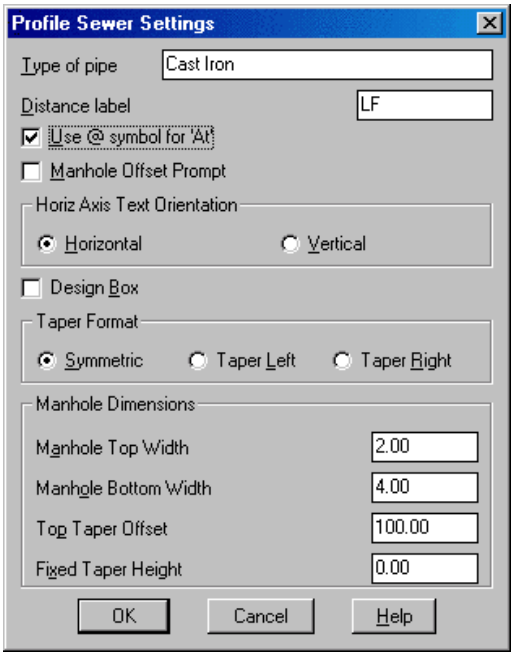


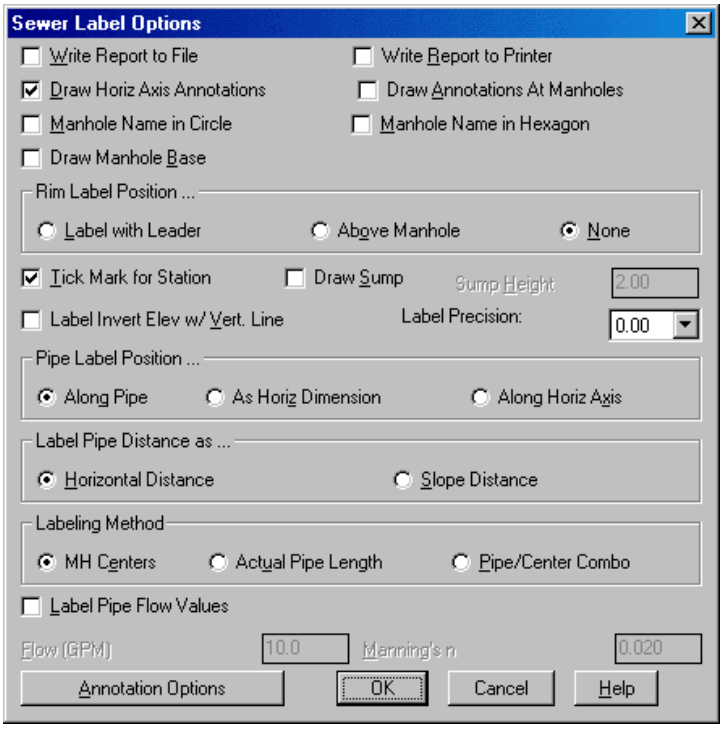
Enter the step up/down in feet <0.00>: *Press Enter*

Size of pipe in inches <10.0>: *8.0*
Station of next manhole or pick a point (U,E,D,Help): *Press Enter*

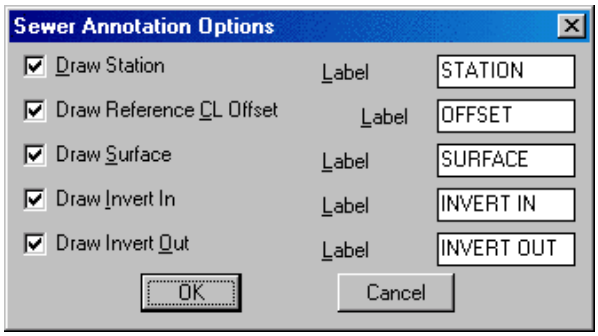
Profile Sewer Settings dialog

Sewer Label Options dialog





Sewer Annotation Options dialog (displayed by pressing the Annotation Options button)



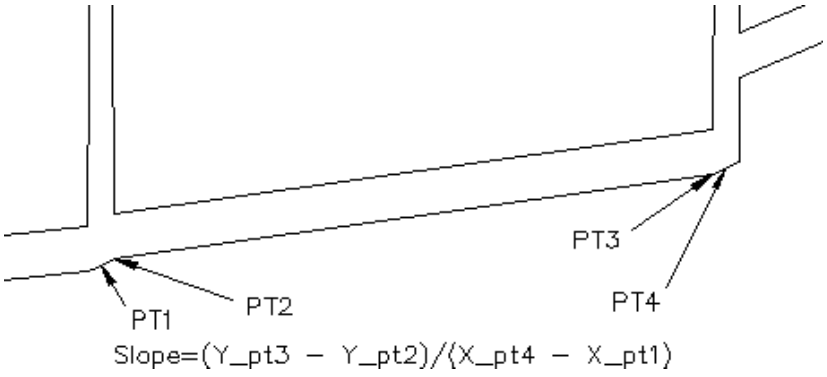
Select existing ground polyline: *Pick a polyline or press Enter to be prompted for each manhole surface elevation.* This prompt only appears if no ground polyline was selected above.

Manhole No. 1 label [MH #1]: *Press Enter*

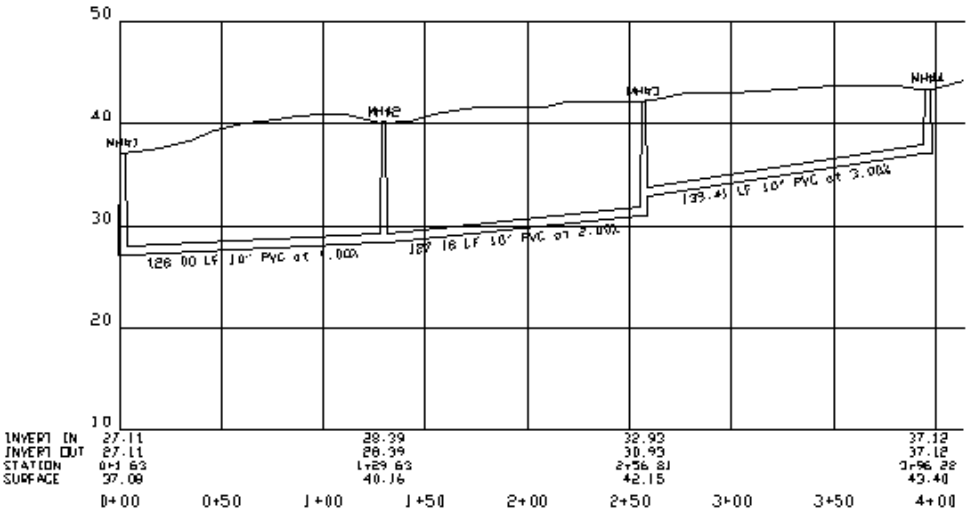
Manhole No. 2 label [MH #2]: *Press Enter*

Prerequisite: A profile grid

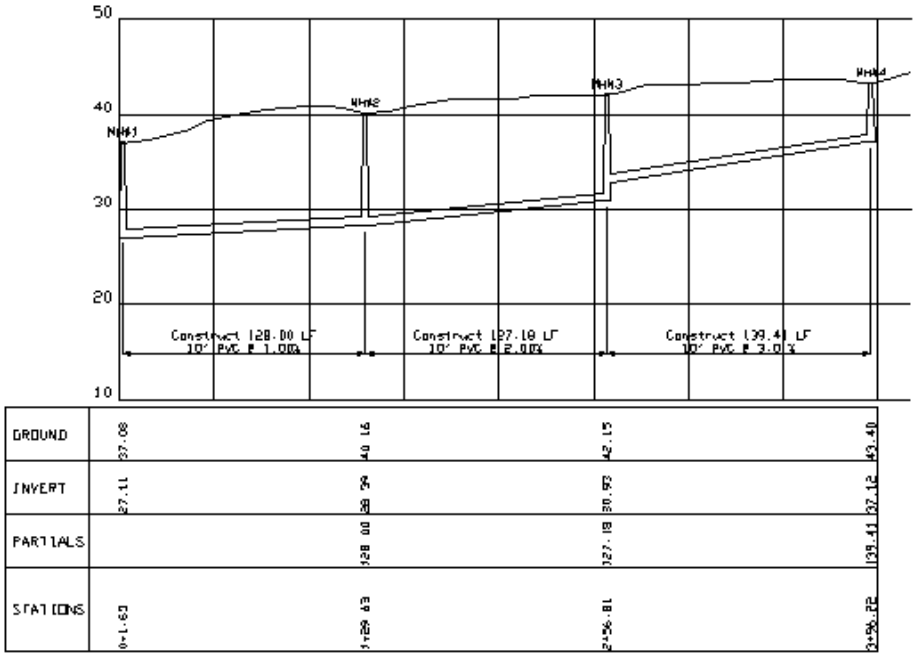
Keyboard Command: sewer



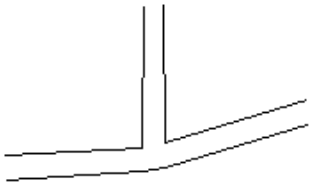
Pipe/Center Combo Labeling Method calculates the slope as the elevation difference from the edge of the pipe divided by the distance between the manhole centers.

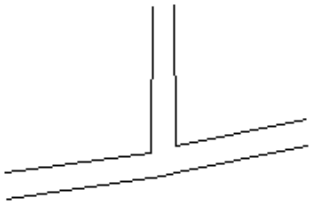


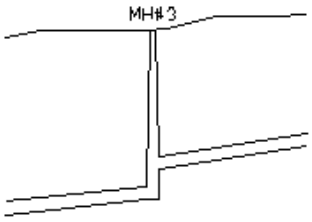
Example of sewer profile and surface profile



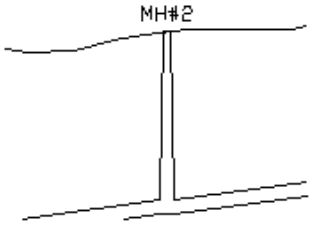
Example of sewer profile using Horizontal Axis Text Orientation as Vertical and Pipe Label Position as Horizontal Dimension

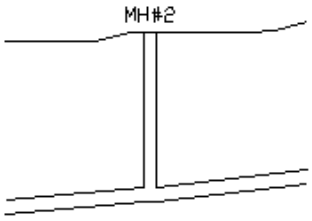


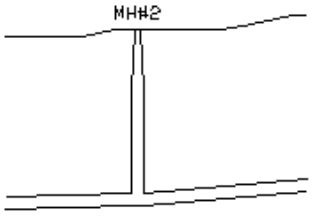




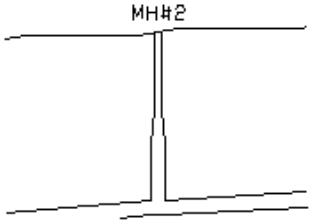
Detail of manhole bottom at pipe slope
Detail of drop across manhole of 0.2
Detail of step up

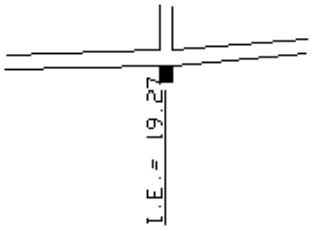


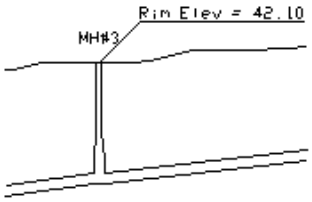




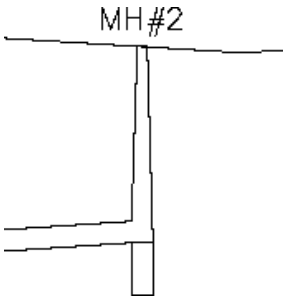
Top=2, Bottom=4, Offset=4, Fixed=0
Top=2, Bottom=4, Offset=100
Top=4, Bottom=4

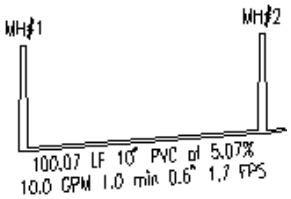






Detail of Label Rim Elevation at Manhole
Top=2, Bottom=4, Offset=4, Fixed=2
Detail of Draw Manhole Base and Label Invert Elevation with Vertical Line



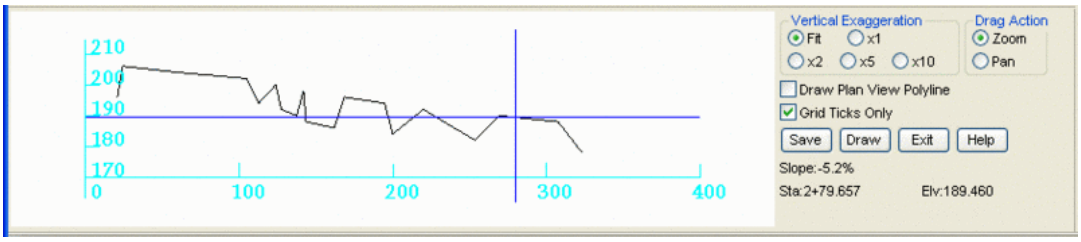


Label Pipe Flow Values option shows flow rate, travel time, depth and velocity
Manhole with the Draw Sump option

Quick Profile from Screen Entities

Function

This command allows you to create, view, and draw profiles from the current surface.



Pick starting point (CL-Centerline, P-Polyline): To make a profile you need to define the alignment by: 1) picking points on the screen; 2) typing in CL in the command prompt, and selecting a centerline file; or 3) typing in P and choosing a polyline from the screen. After doing so, the above profile viewer is created.

Quick Profile from Screen Entities creates a profile from all the entities that cross your alignment. When you move the cursor around the profile viewer a crosshair follows along the surface and reports the Station, Slope %, and Elevation at each point. It is displayed towards the bottom-right side of the screen. In this example the station is 2+79.657, the Slope is -5.2%, and the Elevation is 819.460. A crosshair can been seen in the profile drawing and along the alignment in the main drawing as well.

Vertical Exaggeration: x1 is the actual appearance of the surface(s). Depending on the flatness of the surface(s), you can select x2, x5, x10 vertical exaggerations to better see the elevation differentiation and different surfaces. The option Fit automatically exaggerates the vertical to best fit the profile viewer.

Drag Action: This dialog allows you to zoom in and out, and pan around the profile. To zoom in click and drag up, to zoom out click and drag down. To Pan, click amd drag the direction you want to move.

If you created a profile alignment by picking points and you want to save that polyline you created then toggle on Draw Plan View Polyline. If you do not choose Draw Plan View Polyline than the polyline will be lost when you exit out of the Quick Profile from Screen Enities command. Grid Ticks Only marks elevations and distances but does not draw them into grids.

The Save icon allows you to save the profile as a (.pro) file by whatever name you give it. The Draw icon allows you to draw the profile right on your drawing. Set the layer name, vertical and horizontal scale as desired, pick a starting point to draw, and the profile is created.

Prompts

Command: *tk_quickpro2*

Pick starting point (CL-Centerline,P-Polyline): *P to select a polyline from the screen*

Select profile centerline polyline: *Select the desired polyline*
Tested 39 of 39 Entities Intersects found> 12

Opening file C:/Program Files/Carlson TakeOff 2004/quickpro.pro for write.

Prerequisite: entities

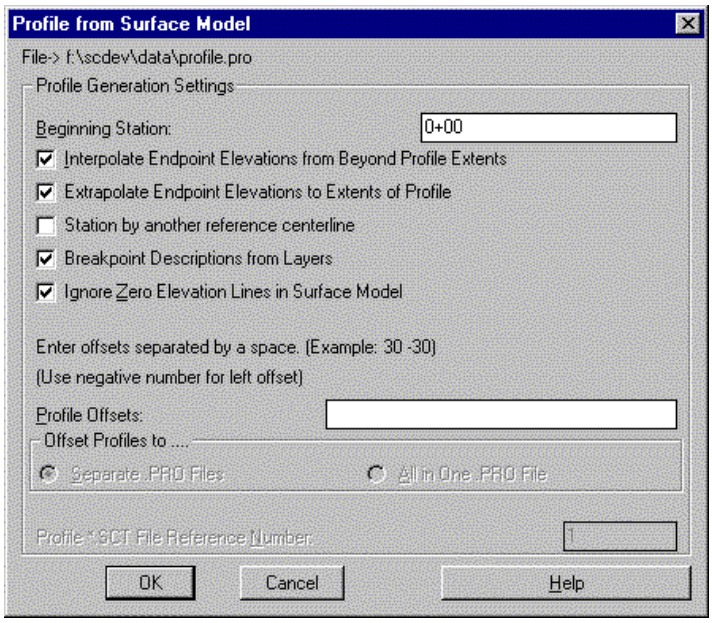
Keyboard Command: tk_quickpro2

Profile from Screen Entities

Function

Profile from Surface Entities creates a profile from contours, triangular mesh, and other 3D drawing entities. The method is to draw a polyline as the profile centerline. Then the profile is derived from the intersections of this polyline with the 3D entities. For added accuracy in pulling the profile, include the triangular mesh as well as the contours.

The Interpolate Endpoint Elevations from Beyond Profile Extents option will cause the program to look past the ends of the centerline for additional intersections with 3D entities.These additional intersections will then be used to interpolate the elevation at the starting and ending station of the centerline. The Station by Another Reference Centerline option will prompt you to pick another centerline polyline. The intersection points along the first centerline are then projected onto the second centerline. The profile then stores the elevation of the intersection with the station along the second centerline.



In addition to creating the one centerline profile, offset profiles may also be created by entering the offset distances in the Profiles Offset box of the Profile from Surface Model dialog.

Prompts

File Selection Dialog

Specify a profile file name to create.

Profile from Surface Model dialog box (previous page)

Polyline should be drawn in direction of increasing stations.

CL File/<select polyline which represents the profile centerline>: *Pick the centerline*

Select Lines, PLines, and/or 3DFaces that define the surface for profiling.

Select objects: *C* (for crossing and window everything the centerline crosses)

Prerequisite: A polyline centerline and surface lines and polylines.

Profile from Grid or TIN File

Function

This command creates a profile (.PRO file) from a centerline polyline and a surface model stored in a 3D grid file (.GRD) or triangulation file (.TIN). The polyline defines the alignment of the profile and the grid defines the surface.

Prompts

Choose Grid or TMESH file to process

Select the 3d grid file or tmesh file.

PROfile file to Write dialog box

Enter a profile file name to write.

Select profile polyline centerline: *Pick a polyline*

Enter the starting station <0.0>: *Press Enter*

Prerequisite: A .GRD grid file or .flt tmesh file.

Keyboard Command: progrid

Profile from 2D Polyline

Function

This command allows you to convert a polyline that is drawn on a profile grid into a profile (.PRO) file. The polyline must be drawn in the direction of stationing.

Prompts

New or Append Dialog Box

Choose New unless you intend to create a multiple profile.

File Selection Dialog Box

Specify the profile (.PRO) file to create.

Profile Settings Dialog

Set these parameters to match the dimensions of the grid.

Pick the lower left grid corner: *Pick the grid corner.* Endpoint snap is set on.

Profile number <1>: *Press Enter.* This is an optional profile name used for multiple profiles.

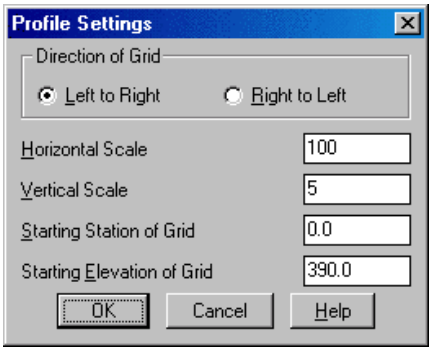
Select the polyline to write profile from:

Select object: *Pick the polyline in the grid.*

A station and elevation report is produced.

Prerequisite: Drawn polyline which represents profile.

Keyboard Command: pro2dpl



Profile from 3D Polyline

Function

To create a .PRO file, Profile from 3D Polyline uses X-Y distances between the points of a 3D polyline for sequential stations and the Z values at these points for profile elevations.

Prompts

[nea on] Select polyline to Station/measure: *pick a 3D polyline*
Profile File to Write dialog Specify a profile file name to create or append an existing file.
Profile number <1>: *press Enter* This is an optional profile name useful for multiple profiles.
Station by another reference centerline [Yes/<No>]? *press Enter*
Starting Station <0.0>: *press Enter* This is the station at the start of the picked or reference polyline.
Prompt for elevations [Yes/<No>]? *press Enter*
The new profile is then stored.

Keyboard Command: pro3dpl
Prerequisite: A 3D polyline

Profile from Points on Centerline

Function

This command creates a .PRO file from TakeOff points and a centerline that is represented by a polyline. The elevations of the profile are derived from the elevation of the points and the stationing for these profile points is calculated from the distance along the centerline. The points must be within the offset distance from the polyline in order to be included in the profile. The polyline should be drawn in the direction of increasing stations.

Prompts

PROfile file to Write dialog box Enter a profile file name to write.

Select centerline polyline: *pick a polyline*

Enter the centerline's starting station <0.0>: *Press Enter*

Enter the maximum offset tolerance <1.0>: *Press Enter*

Ignore zero elevations (<Yes>/No)? *Press Enter.* The points should be located on the real Z axis.

Select the Carlson points along the centerline.

Select objects: *select the point entities*

Prerequisite: A polyline centerline and Carlson point inserts

Keyboard Command: profpts

Import Profile

Function

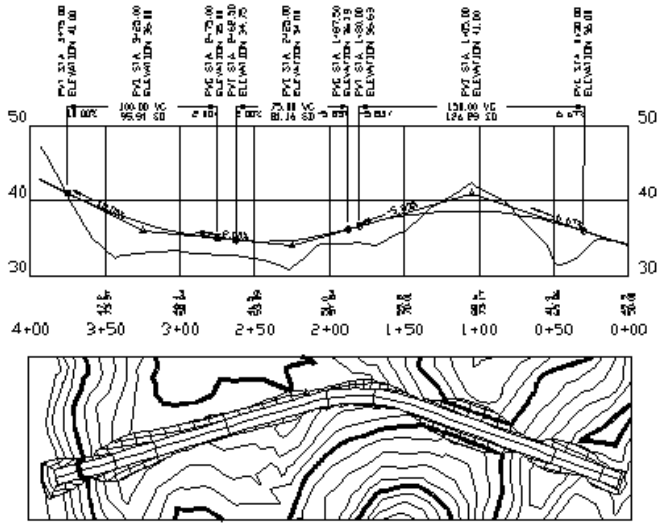
This command converts Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, and Leica road files into Carlson TakeOff profile (.PRO) files.

Prerequisite: a Terramodel, Geodimeter, GeoPak, Sokkia/Leitz, Softdesk, or Leica road file

Profile To 3D Polyline

Function

This command converts a 2D polyline centerline into a 3D polyline that follows the elevations of the profile. Horizontal and vertical curves are represented as a series of polyline segments since 3D polylines cannot contain arcs.



Example of road design in plan-view with Profile to 3D Polyline

Prompts

Layer for 3D Polyline <3DPROF>: *Press Enter*

Select profile centerline polyline: *pick a polyline*

Enter the starting station <0.0>: *Press Enter*

Select Profile File

Erase centerline (Yes/<No>)? *Press Enter*. This option will erase the original 2D polyline centerline.

Prerequisite: A .PRO file and a centerline polyline

Keyboard Command: proto3dp

Profile To Points

Function

This command creates Carlson TakeOff points along a horizontal alignment polyline using a profile file to compute the point elevations. The created points are stored in a coordinate (.CRD) file and can also be drawn on screen in the layer specified by the user. Station text, profile name, and special points (vertical and horizontal PC's and PT's) can be stored in the point description depending on user settings.

Create points at Profile special points: Includes vertical PC and PT points.

Create points at Centerline special points: Includes horizontal PC and PT points.

Create points at Station Intervals: Allows you to specify intervals for point creation.

Interval On Line Segments: Specify station interval for line segments.

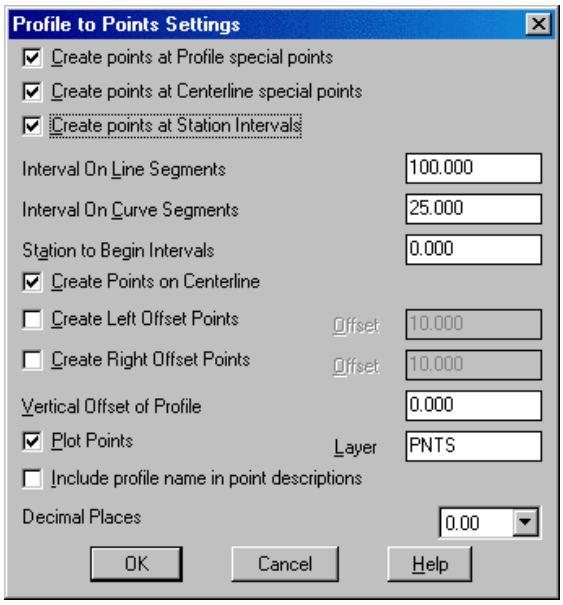
Interval On Curve Segments: Specify station interval for curve segments.

Station to Begin Intervals: Specify station to start intervals.

Create Points on Centerline: When checked, points will be created on the centerline.

Create Left Offset Points: When checked, left offset points will be created. Specify the offset in the edit box.

Create Right Offset Points: When checked, right offset points will be created. Specify the offset in the edit box.



Vertical Offset of Profile: Specify the vertical offset. Enter zero for no vertical offset.

Plot Points: When checked, points will be plotted in the drawing, otherwise points are only added to the current coordinate (.CRD) file.

Include profile name in point descriptions: When checked, the profile name will be used as the prefix on the the point description. For example, if the profile name is DESIGN.PRO, then the point description might be DESIGN 0+63.37

Decimal Places: Specify the display precision for points that are plotted in the drawing. This setting does not affect the coordinates stored in the CRD file.

Prompts

Select Coordinate File to Process

If the current coordinate is set, it is used automatically without this prompt.

Select profile centerline polyline: *pick a polyline*

Starting station of centerline <0.0>: *Press Enter*

Choose Profile to Process dialog

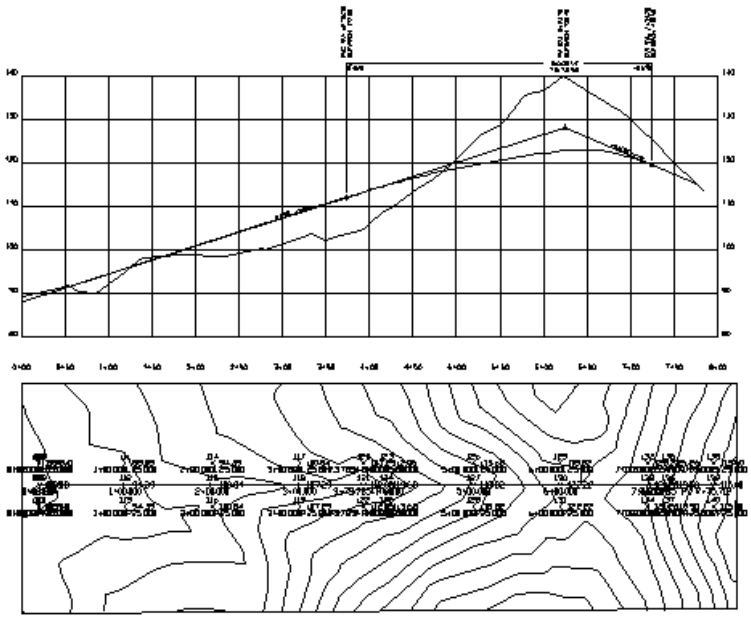
Specify a profile name.

Starting point number <1>: *Press Enter*. This defaults to the point number after the highest one currently in the CRD file.

Station for additional point (ENTER to end): *Press Enter.* This option will create a point at the specified station.

Prerequisite: A .PRO file and a centerline polyline

Keyboard Command: pro2pts

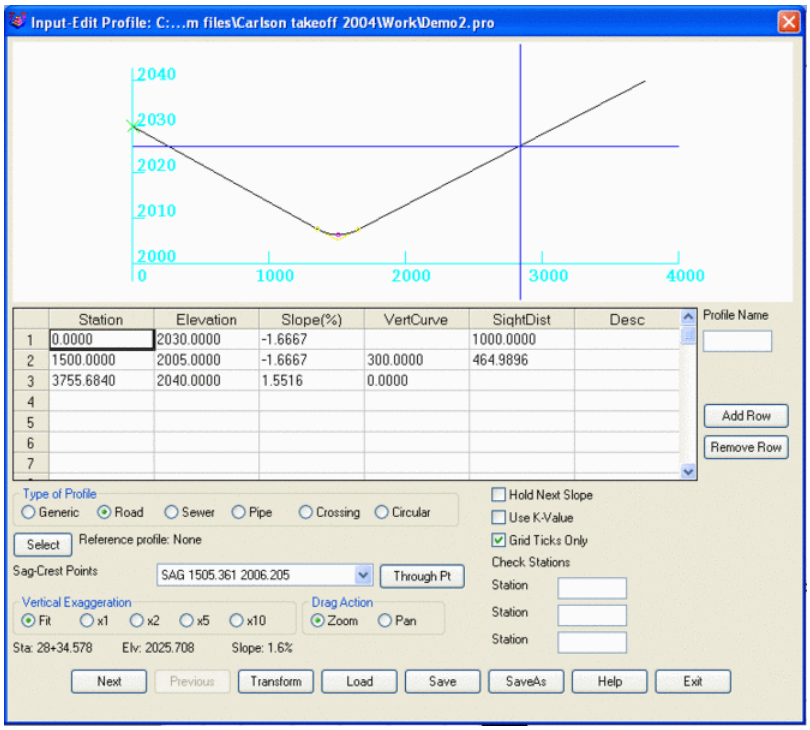


Points created along profile centerline using elevations from the above road profile

Input-Edit Profile File

Function

This command is a spreadsheet type editor for profile (.PRO) files. Besides editing a profile, this routine can be used to just view the contents of a profile. Also, a new profile can be entered by editing a previously empty or nonexisting file. This tool includes a user-interface that has a graphic window to dynamically show the profile during editing or creation. This Routine also includes a real-time report of station-elevation-slope as the cursor is moved along the graphic view of the profile.



To create a new profile by manual entry, first choose what type of profile to create. Profile types are either Generic, Road, Sewer, Pipe or Crossing. For each type, different data entry options are available and will display in the data editor when selected. For example, a Generic profile has the following headers, Station, Elevation, Slope and Desc. A Road profile will have the following headers in addition to the headers for the Generic profile, VertCurve & SightDist. Each profile type has data entry fields and Headers for the data specific to the profile type. Next enter the Station, PI Elevation or Slope or the segment in the appropriate field. You may change data fields using the Enter Key, the Tab Key or by clicking in the desired data field with the left mouse button. As data is entered, the profile will be drawn in the graphic window.

To edit Profile data, highlight the row of profile data to edit by clicking on it with the left mouse button. Next select the data to edit. After making the necessary change to the data, to accept the change press the enter key, tab key or click in another data field using the left mouse button. The changes to the profile with be displayed in the graphic window.

To review/inspect the profile data using the real-time report, simply move the cursor along the profile in the graphic window. The Station, Elevation and Slope of the segment is displayed on the dialog box to the right of the data editor.

Vertical Exaggeration can be applied to the graphic display by selecting a value from the Vertical Exaggeration options.

Drag Action: The graphic display has zoom and pan capabilities. Choose how the display reacts to pressing and holding the left mouse button by selecting either Zoom or Pan. With Zoom selected, while holding the left mouse button in, dragging in the direction of the (-) will zoom out. Dragging in the direction of the (+) will zoom in. With Pan selected dragging in any direction will pan the display in that direction.

On the right is a column for **Check Stations** which report the elevation at the specified stations. The Check Stations are not stored in the profile. This is a design tool for viewing the elevations at certain stations while adjusting the profile data. The last line has eight action buttons.

Insert: Adds a new row into the profile after the current row.

Delete: Removes the current row.

Transform: Allows you to either Translate or Scale the profile. Translate globally adds or subtracts value to stations and/or elevations within the specified range of stations while Scale will apply the specified scale factor to stations and/or elevations within the specified range of stations.

Next: Used for navigation when editing a .PRO file containing multiple profiles, loads the next

profile.

Previous: Used for navigation when editing a .PRO file containing multiple profiles, loads the previous profile.

Profile Number: Displays the sequential number of the active profile for multiple profile files.

Use K-Value: Toggles between displaying K-Value and Sight Distance in the fifth column for road profiles.

SaveAs: Allows you to save the profile under a different profile file name.

Save: Saves the profile using the current profile file name. The current profile file name is displayed in the top title bar of the dialog box.

Quit: Exits the routine without saving anything.

Prerequisite: None

Keyboard Command: profedit

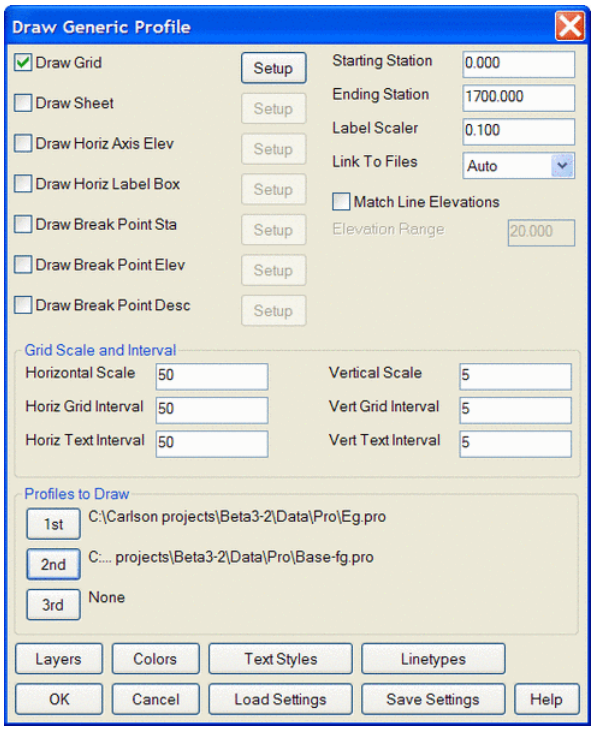
Draw Profile

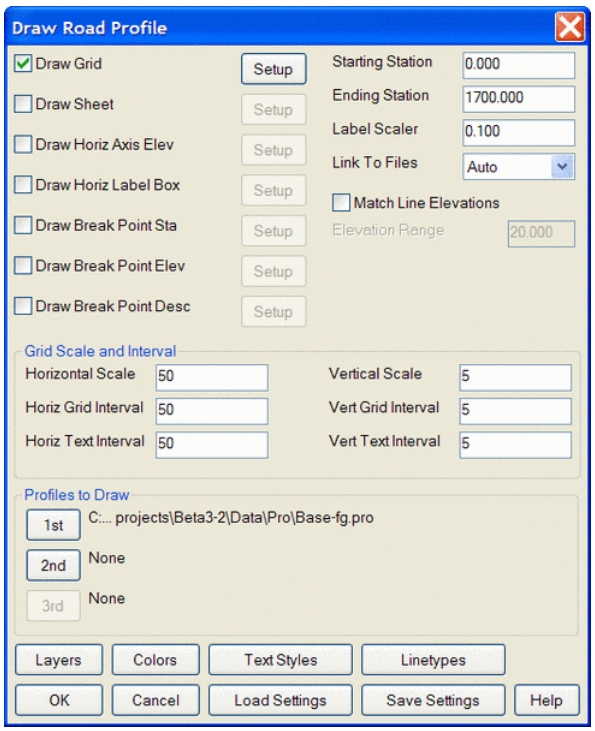
Function

Draw Profile is a flexible routine for drawing a profile anywhere in the drawing. The profile may be drawn with or without a grid or with just tick marks. The vertical curve annotations, for a road profile, and manhole annotations, for a sewer profile, may also be drawn. Draw Profile uses the profile information that is stored in .PRO files. Once the profile is drawn using Draw Profile, the design and labeling routines of the Profiles dropdown are applicable to the profile. The first step in Draw Profile is to choose a profile (.PRO) file.

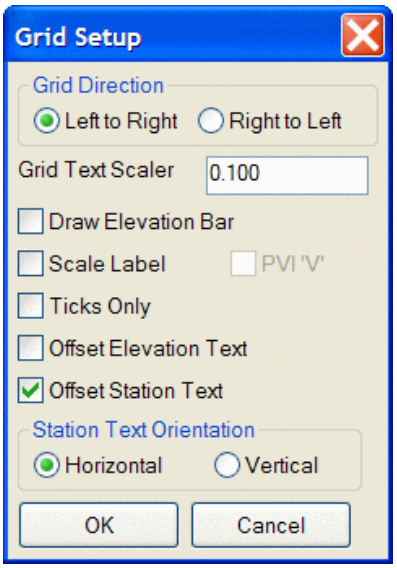
If a profile of a Surface is selected, the Draw Generic Profile dialog is presented, however if a profile of a proposed road design is selected, the Draw Road Profile dialog is presented.

If the Draw Road Profile dialog is used, or if a proposed road design Profile is added as 2nd or 3rd Profile to Draw in the Draw Generic Profile dialog, a subsequent dialog is presented specifically for the settings for drafting Vertical Curves. If no road design profile is involved, the Vertical Curve settings dialog is not invoked. If a Pipe profile is selected, a special dialog with settings for drafting pipes and structures is presented.

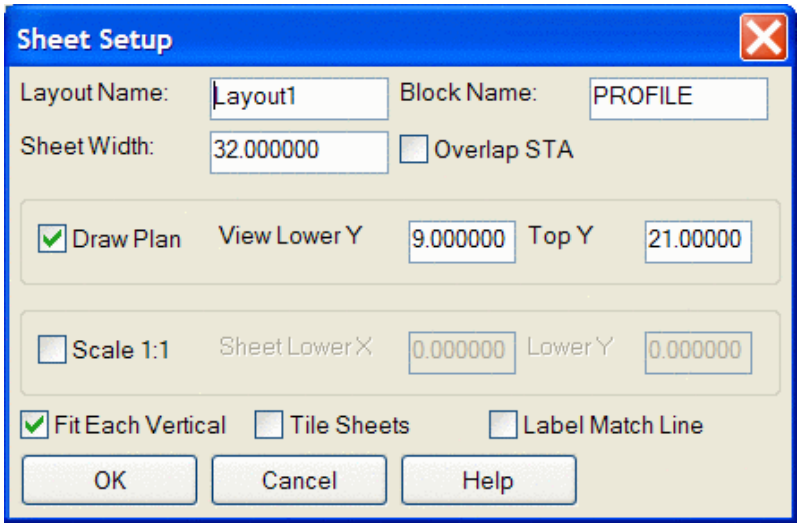




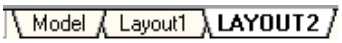
Draw Grid: This option will draw a grid and axis elevations for the profile.



Scale Label: Click on this option and you obtain a scale drawn at the lower left corner of the profile.
Grid Direction: Profiles can be drawn Left to Right (the default) or Right to Left. Although most profiles are drawn left to right, if you have a road that runs east to west and you wish to draw the profile stationing beneath the actual road stationing, then choosing a Right to Left profile may be appropriate. Unavailable when Draw Sheet is checked.
Draw Sheet: When checked, the profiles will be drawn in paper space. When used along with the Draw Plan options within Sheet Setup, plan and profile sheets can be created. In AutoCAD 2000 and later, each plan and profile sheet is created in it's own layout tab. When the plan and profile is drawn, you are placed in tilemode=0 and paper space. Click the "model" space tab (shown below) to return to model space to edit the plan view features, for example. The options within Sheet Setup become available when this toggle is checked on. Here is the Sheet Setup dialog:



Layout Name: Enter a name for the paper space "tabs" to be assigned to each layout for each sheet. The program will automatically divide the plan view (if "draw plan" is clicked) and the profile view into sheet layouts, and if the length of the profile extends beyond a single sheet, then multiple layouts are created, with the layout name ID incremented by 1. If you enter "ms" to go to model space within a layout tab, you can pan to alter the plan view position. Its best to zoom in/out and edit within the Model tab. The Layout tabs appear at the bottom of the screen, along with the "Model space" tab to go back to standard plan view:



Block Name: This is the drawing name for the plan and profile sheet to be inserted. SurvCADD provides a standard plan and profile drawing in the form of Profile.dwg located in the Support subdirectory. You may wish to revise Profile.dwg, add your company logo, and re-save it as Profile1.dwg or you could add your own complete version of a Plan and Profile sheet. If you choose the latter, you should examine the scale, dimensions and the lower left corner of Profile.dwg, and try to duplicate those dimensions and corner coordinates in your own drawing. It is also important to store all your standard profile sheets in the Support subdirectory. You cannot draw Right to Left in Sheet mode. Note that the Sheet mode will re-orient the centerline left to right, which may cause text (such as the stationing) to plot upside down, until you use the *Flip Text* command under the Edit menu.

Sheet Width: This is the profile width, in inches, on the sheet. Even though the sheet is a fixed size, you can limit the length of the plot to 32 inches or less with this entry. If we used an entry of 16 for the example profile above, two plan and profile sheets would be created, because the first sheet would go from station 0+00 to 3+20 (16 inches at a 20 scale) and the second sheet would finish from 3+20 to 4+51.91. In English units, a typical entry here would be 30 for 30 inches.

Overlap Station: In multiple plan and profile sheet plotting, after the first sheet, all subsequent sheets will have the first 2 stations in common with the last 2 stations on the previous sheet, if the Overlap Station option is turned on. For example, if the last 2 stations are 3+10 and 3+20 on sheet 1, then sheet 2 will start with 3+10, then 3+20, with this option turned on. With this option turned off, if the first sheet ends with 3+20, then the second sheet would begin with 3+20.

Draw Plan: With this clicked on, you are asked to select the polyline that represents the centerline, and the program best fits the centerline in the plan view portion of the plan and profile sheet, then captures all of the associated drawing that will fit in that paper space window. If the length of the polyline divided by the scale exceeds the Sheet Width entry (for example, 5000 feet of road divided by 50 Horiz. Scale is 100, which exceeds the sheet width), then multiple plan and profile sheets will be automatically created. A 5000 foot road at 50 scale with a 30-inch sheet width, would lead to 3 full sheets of 1500 feet each and a "leftover" fourth sheet showing the last 500 feet.

View Lower Y: This sets the lower position of the paper space window for the plan view. With Lower Y set to 9 (inches above the base of the sheet) and Top Y set to 21, there is a 12 inch vertical window, running the full Sheet Width (typically 30 to 32). This window for the plan view can be expanded or reduced with these settings.

Top Y: This sets the top vertical limit for the plan view window, measured in inches from the bottom of the plan and profile sheet.

Scale 1:1: With this clicked off (the default), the profile will draw in scaled units (e.g. 1"=50'), with a 1500-foot profile measuring 1500 feet. With this clicked on, the units will be 1:1. A 30-inch profile sheet will measure 30 units, even though the centerline and profile may be 1500 feet in length. If the Scale 1:1 option is turned on, then you cannot check the distances of features using commands such as Bearing and Distance on the Inquiry menu, because the distances will be scaled down by a factor equal to the drawing scale (for example, at 1"=50', the reduction in scale factor is 1/50 or 0.02). You can set the absolute starting coordinate for the 1:1 scaled plot by setting the Sheet Lower X and Sheet Lower Y.

Fit Each Vertical: With this option turned on, the command will recognize the lower and upper vertical elevations of the profile and set the vertical axis elevation range to enclose the actual elevation limits of the profile. With this option turned off, you can enter the lower vertical elevation range, dropping it down further to increase the lower margin. Whether or not the Fit Each Vertical option is turned on or off, you are always prompted for the top elevation range.

Tile Sheets: If clicked on, only one Layout is created in paper space, and all sheets appear in this single Layout as tiles of individual sheets, much like the tiles mode of viewing files within Windows Explorer.

Label Match Line: When clicked on and multiple sheets are plotted with plan view option on, a match line will

plot in the plan view.



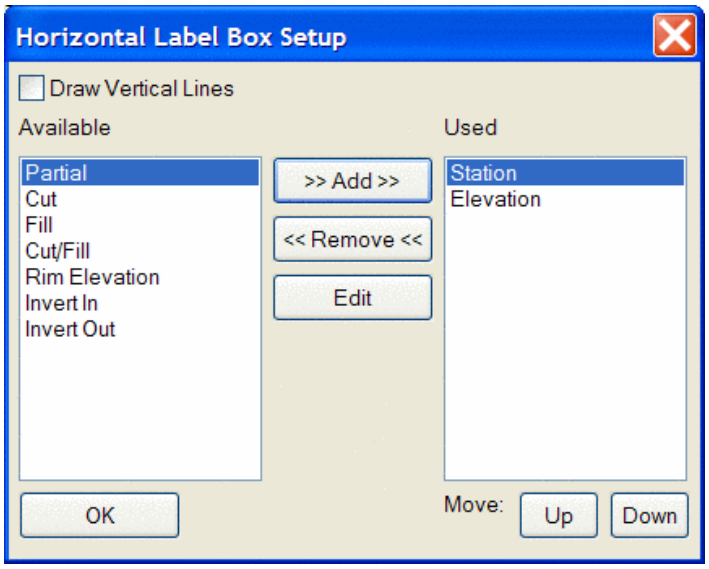
Prompts (Draw Sheet option)

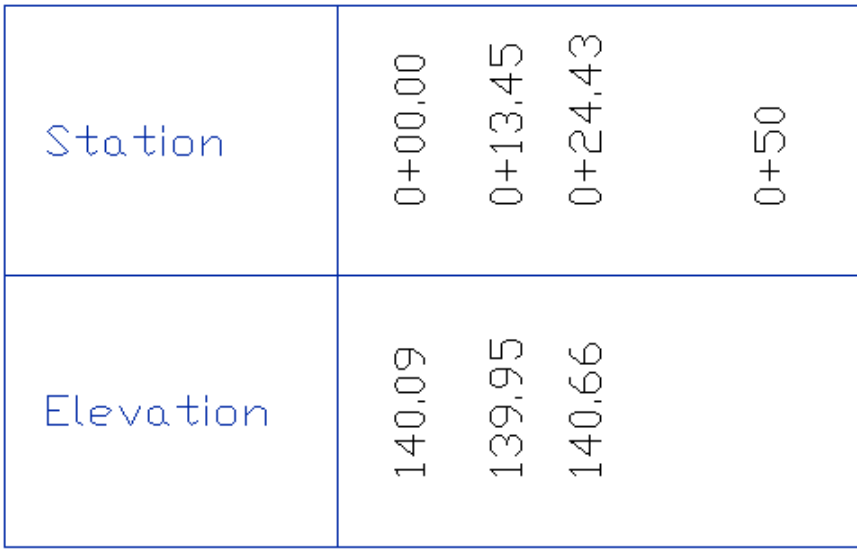
Bottom Vertical Spacing <0.0>? *press Enter*

The program places the profile plot flush against the bottom of the vertical grid, by default. This prompt allows for an offset, moving the profile plot up off the bottom of the grid. If the lowest elevation of the profile is 940 by default, entering 10 would start vertical axis labeling at 930, and if the vertical scale was 5 units, this would push the vertical plot up 10 units or 2 standard grid intervals.

Top Elevation of Profile Grid <945.0>: *press Enter*
Cl File/Select polyline that represents centerline: Select the centerline polyline (if Draw Plan has been clicked on).
Beginning Station <0.0>: *press Enter*
The items below refer again to the profile options that are independent of the Draw Sheet option:

Draw Horz Label Box: This option draws a boxed area underneath the profile. It is best used in standard Draw Grid mode, with Draw Sheets clicked off. An example of the resulting plot is shown here:





Decimals: The default is 3 decimal places, typical of metric accuracy (1 mm). This is applied to the stations, partial distances, and elevations of the break points on the profile. This decimal setting is in association with the Horizontal Label Box option.

Vertical Lines: This option enables or disables the drawing of the vertical lines that appear in the graphic above, also in association with the Horizontal Label Box option.

PVI 'V': This option plots a special "V" look above all vertical curve PVIs (points of vertical intersection).

Draw Break Pt Elev: This option draws elevations vertically above all break points on a profile. It is typically applied to generic ground profiles and not to road profiles.

Decimals (for Break Points): Typically a setting of 1 or 2 decimals is applied to break points for existing ground.

Text Scaler (for Break Points): This text scaler is multiplied by the Drawing Scale set in the Drawing Setup command under the Settings menu, to determine the actual text height. For example, if the horizontal scale is set to 50, a text scaler setting of 0.1 will produce a text height of 5.

Ticks Only: If this option is selected, full grid lines are not drawn. Instead, a fringe of ticks are drawn along the

two vertical axes and the horizontal axis.

Match Line Elevations: For high relief profiles that might otherwise extend up and into the plan view portion of the drawing, the Match Line Elevations option can be used to break the profile and redraw the remaining portion with its own vertical scale, as seen above.

Elevation Range: This is the range of elevations that is used in conjunction with the Match Line Elevation option. If the range is exceeded (that is, if the range above is 20), the program will break the profile and draw the remainder with a separate vertical axis range.

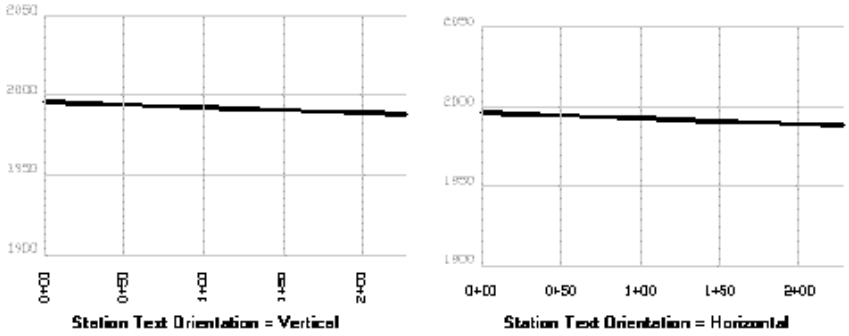
Draw Break Pt Desc: Similar to the Draw Break Pt Elev option, this option draws vertically, at each break point, the description associated with the profile points. If a profile is made by the Profile from Surface Entities command, there is an option to store the layer of any polyline used for interpolation in the description for the profile break point. In addition, within the Input-Edit Profile command, you can enter a description. The descriptions are plotted with this option turned on.

Design Box: This is very similar to the Draw Horizontal Label Box option discussed above. This draws three rows of information beneath the profile, with text plotted vertically. The first row shows the elevation of each vertex or break point in the profile. The second row shows the "partial" or incremental distance between break points, and the third row shows the station of each break point. This command does not draw any line work associated with these rows of text.

Offset Station Text: When clicked on, this option offsets the horizontal axis text by an amount equal to about 6 standard text characters, allowing the insertion of elevation or other information above the stationing. It is often used in conjunction with the Label Horizontal Axis options.

Offset Elev Text: This option offsets the left-side vertical axis text a distance equal to the horizontal scale.

Station Text Orientation: This option allows you to specify the orientation of the station text shown along the bottom of the profile. The example below shows both options:



Horizontal Scale: This scale applies primarily to text size. If the text scaler is 0.1 and the horizontal scale is 50, then text size will be $0.1 * 50 = 5$.

Horizontal Grid Interval: This sets the spacing of the grids that run vertically from the horizontal scale.

Horizontal Text Interval: This sets the spacing of the stationing text that appears along the horizontal axis. When using a large "Axis Text Scaler", the horizontal axis text can become too large, and it often necessary to space the horizontal text interval at twice the horizontal scale.

Vertical Scale: This scale sets the vertical exaggeration of the profile. If the horizontal scale and vertical scale are the same, then the vertical is not exaggerated. Profiles are often plotted with a 5 or 10 vertical exaggeration. For example, the horizontal scale may be 50, but the vertical scale may be 5.

Vertical Grid Interval: This sets the spacing of the grids that run horizontally between the vertical axes on the left and right side of the profile.

Vertical Text Interval: This sets the spacing of the elevation text that appears along the vertical axes.

Axis Text Scaler: This sets the size of the horizontal and vertical axis text to the horizontal scale times the scaler, when you are working in English units. In metric units, the text height would be 0.01*horizontal scale*scaler. In English units, a scale of 0.2 or more will usually require that the Horizontal Text Interval be set to twice the horizontal scale (every other inch).

Label Text Scaler: This sets the size of text used for vertical curve annotation to the horizontal scale times the scaler, when you are working in English units. In metric units the text height would be 0.01*horizontal scale*scaler.

Starting Station: This field defaults to the starting station in the selected profile(s). If changed, the starting station can move forward, clipping out the first part of the profile. When you are not plotting sheets, you must set the starting station to the end of the previous sheet's ending station to force a multiple sheet layout.

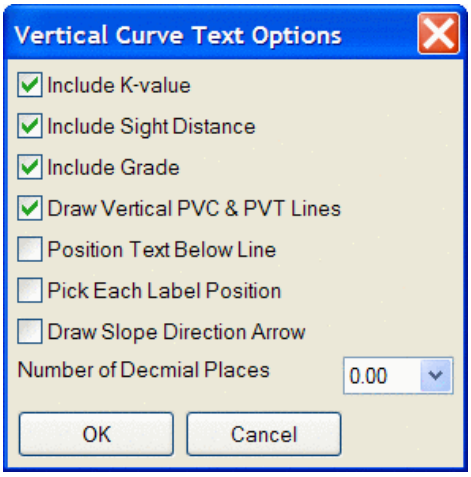
Ending Station: This field defaults to the ending station in the selected profile(s). A profile that is 3000 feet in length could be plotted in 2 parts, first station 0 to 1500, then station 1500 to 3000, using the Starting Station and Ending Station options.

Profiles to Draw: This portion of the Draw Profile dialog box allows up to 3 profiles to be selected and plotted simultaneously. These profiles can be distinctly layered using the Layers button at the base of the dialog box. If the goal is to plot more than 4 profiles on the same drawing, run the Draw Profile command a second time, and specify up to 3 more profiles, choosing the same scaling and lower left corner for placement. Be sure to turn off Draw Grid when placing additional profiles on a pre-drawn profile. Otherwise the grid and axis text may be drawn more than once.

When OK is clicked at the base of the dialog box, the prompting at the command line continues. In this example, assume that a road profile has been selected, since more prompts will occur with road profiles than with generic profiles.

Prompts (Road profile example)

Erase existing profile from drawing [<Yes>/No]? *N* This prompt appears only if you have previously drawn the profile.
Pick Starting Point for Grid <8779.55 , 5716.36>: *pick a point for the lower left corner of the grid*
Assuming a road profile has been selected, the following dialog box appears:



K-value: This option is a function of the change in slopes on either side of the point of vertical intersection.
Sight Distance: This option is computed by the delta slope and is a function of whether the vertical curve is a sag

or a crest.
Include Grades: This option draws slopes along the tangent portion of the vertical curves, with slope direction arrows.
Draw Vertical PVC and PVT Lines: This option draws vertical lines emanating from the PVC and PVT of all vertical curves.
Position Text Below Line: This option draws the PVC, PVI, and PVT information under the picked location for the vertical curve labeling.

Pick Each Label Position: If there were more than one vertical curve in the profile, this option allows you to pick a vertical position for each of the vertical curve's annotation.
Draw Slope Direction Arrow: Draws an arrow to indicate slope direction.

Number of Decimal Places: Choose the decimal precision used in the elevation and stationing annotation for vertical curves.

Next, select a point vertically that corresponds to the position of the left-right lines under which is written percent grade and above which is written the vertical curve length, sight distance, and K-factor, if requested. The PVC, PVI, and PVT stations and elevations are written above or below this picked point depending on dialog box settings.

Additional Prompting for Multiple Profiles

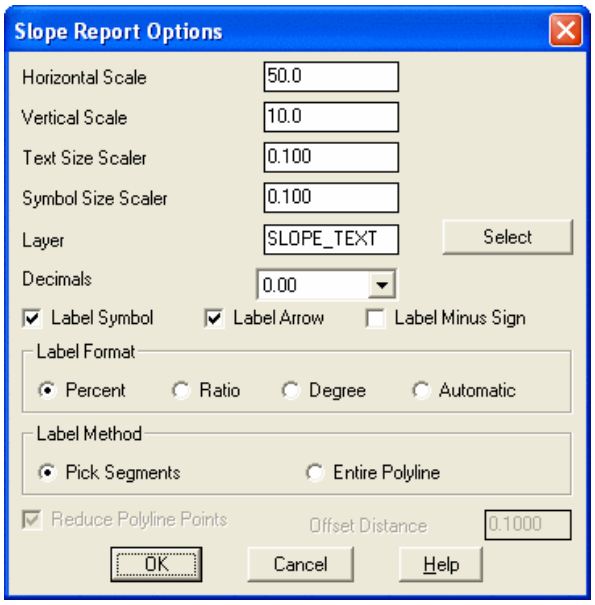
Detected multiple profiles within C:\CARLSON PROJECTS\2006-0124\DATA\PRO\EXAMPLE.PRO
Note that the Profile from Surface Entities command can store additional profiles into the same profile file.
Draw profiles on same or different grids (Same/<Different>)?*press Enter* This determines whether the multiple profiles will be drawn together on the same grid or drawn on separate grids.
Uniform or variable grid size (Uniform/<Variable>)?*press Enter* This selects between one-size-fits all grids or individually sized grids for each profile.

Keyboard Command: drawprof
Prerequisite: A .PRO file

Polyline Slope Report

Function

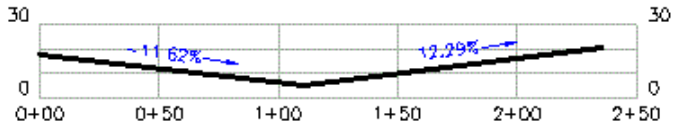
This command calculates and labels the slope of a line, polyline segment, an entire polyline, or pair of points, as drawn on a profile. The command starts with the Slope Report Options dialog.



Horizontal Scale: Specify the horizontal scale of the profile.
Vertical Scale: Specify the vertical scale of the profile.
Text Size Scaler: Specify the text size scaler.
Decimals: Specify the display precision for the slope labels.
Label Symbol: When checked, the degree symbol or percent sign will be used in the label.
Label Arrow: When checked, a slope direction arrow will be included.
Label Minus Sign: Will label a minus sign on negative slopes.
Label Format: Specify how to label the profile slopes. The automatic settings means to use a percent label for any slope less than 10%. and a ratio for any slope greater that 10%.
Label Method: Choose to label the entire profile at once or to pick individual segments.
Reduce Profile Points: When checked, the number of labels created on the profile will be reduced based on the Offset Distance value. Applies only to the Entire Polyline selection option..
Offset Distance: Specify maximum offset between profile vertices. Only available when Reduce Profile Points toggle is checked on.

Prompts

Slope Report Options dialog box
Points/<Select line or polyline to list-label>: *pick a polyline*
Slope Distance> 600.33 Horizontal Distance> 600.00
Elevation Difference: 20.00 Slope Ratio: 30.00:1 Slope Percent: 3.33
Starting point of label ([Enter] for none): *pick a point*
Points/<Select line or polyline to list-label>: *press Enter* If you choose P for points, you go into the Points mode and can label the slope of any pair of screen picks on the profile.



Keyboard Command: llg

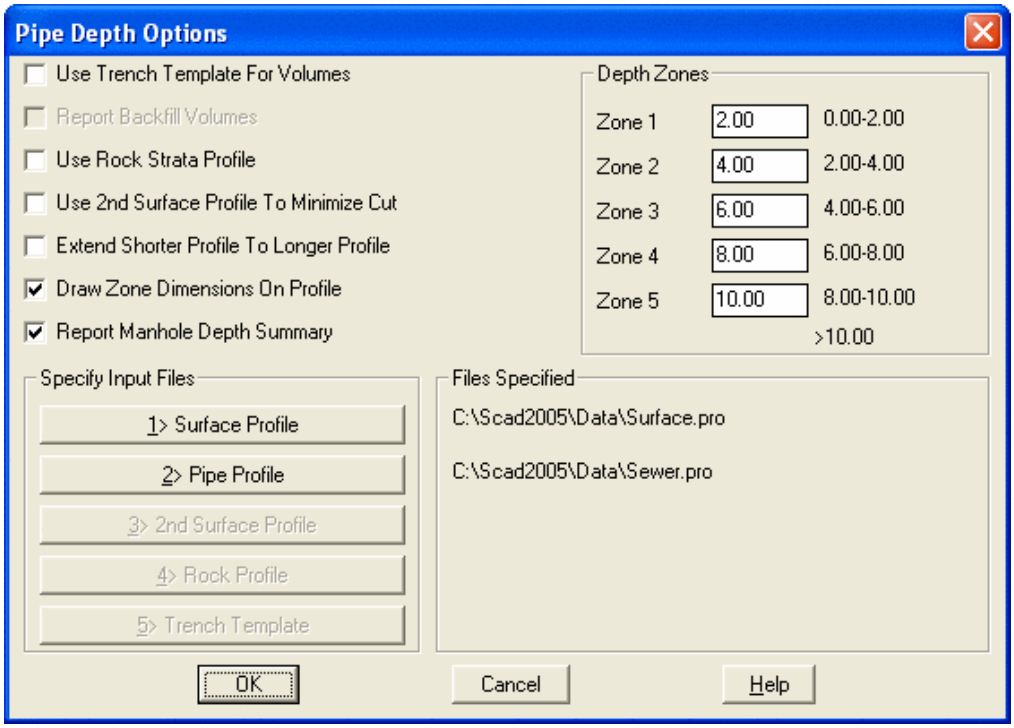
Prerequisite: A profile grid and profile polyline

Pipe Depth Summary

Function

This command reports the horizontal distances for the range of depths comparing a surface profile to a pipe or sewer profile. There is an option to use two surface profiles and the program will use the minimum of the two depths. In addition to the report, the depth ranges can be labeled along the profile in the drawing.

The simplest of applications of this command, comparing a sewer profile to a surface profile and reporting the depth summary according to the specified Depth Zones, is shown below.

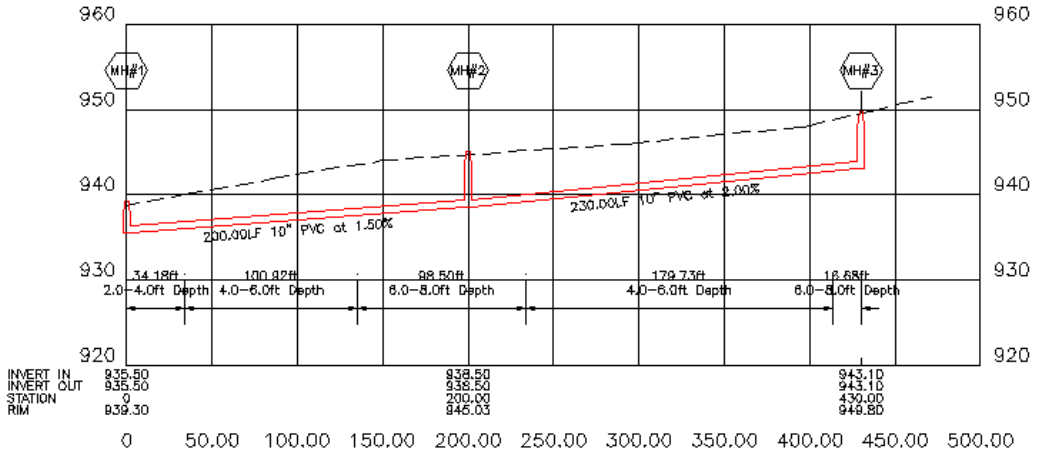


Use Trench Template for Volumes: Trench templates are made using the command Input-Edit Trench Template within the Profile Utilities "flyout". Pipe volumes are then computed.
Report Backfill Volumes: Available if trench templates is clicked on.
Use Rock Strata Profile: If clicked on, the Rock Profile can be entered in the lower portion of the dialog, and if the pipe invert is below rock surfaces along any segment, rock linear feet will be reported, in the same depth categories as used for trench depths. In the example shown below, if rock depth is uniformly 5 feet below surface elevation, in the form of a rock profile, rock quantities are 348 feet of 0-2 feet depth of rock trenching.
Use 2nd Surface Profile to Minimize Cut: If the final grade is below existing grade, in those areas, it saves trenching work to first do the cut to final grade, prior to filling over existing grade in areas of fill. Then trench depths are minimized. This option, if clicked on, computes trench depths to the minimum of the two specified surfaces, and activates the 2nd Surface Profile option in the lower portion of the dialog.
Extend Shorter Profile to Longer Profile: This option will extrapolate the starting and ending stations of the shorter profile to match the longer profile.
Draw Zone Dimensions on Profile: The depth zones will be annotated on the profile drawing with this option.

Report Manhole Depth Summary: This leads to the depth summary report.

Prompts

Pipe Depth Options dialog
Pick lower left grid corner [int on]: *pick the profile grid corner*
Pick vertical position for dimensions: *pick a point below the profile grid*



Pipe Depth Summary

Surface Profile: C:\SCAD2005\DATA\SURFACE.PRO
Pipe profile: C:\SCAD2005\DATA\SEWER.PRO

Depth	Manholes	Linear Ft	% of Total
0-2.0	0	0.0	0.0
2.0-4.0	1	34.2	7.9
4.0-6.0	0	280.6	65.3
6.0-8.0	2	115.2	26.8
8.0-10.0	0	0.0	0.0
>10.0	0	0.0	0.0
Total	3	430.0	

Manhole	Depth
MH#3	6.45
MH#2	6.21
MH#1	3.28
Total:	15.94

Keyboard Command: pipedeep
Prerequisite: Two profiles, one for the surface and one for the pipe invert elevation

Profile Report

Function

This command creates a summary report of generic, road, crossing, pipe and sewer profiles using a profile file (.PRO file). The report is generated in the standard report viewer which can print the report, save it to a file or draw it on the screen.

For roadway profiles, Report Sag and Crest Stations will calculate and report sag and crest stations and elevations. Report Stations at Centerline Points will prompt the user for a centerline file (.cl file) and report stations and elevations at horizontal PC and PT points. Report Cut/Fill from Second Profile will compute and report the elevation difference between the subject profile and a second reference profile. Report Station/Elevation at Interval will calculate and report stations at the specified interval in addition to other points. Report Elevation to Vertical Offset creates and additional elevation column in the report. The differential amount for this column is specified by the user in the Vertical Offset window.

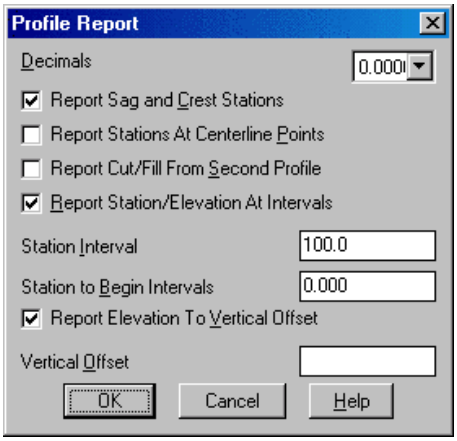
Prompts

Specify a Profile Choose the .PRO file

Profile Report Dialog

Prerequisite: A .PRO file

Keyboard Command: preport



Profile Report

Road Profile

Station Elevation Type VertCurve Distance Slope Desc

0+02.30 88.08 0.00

1+00.00 94.39 6.45%

2+00.00 100.84 6.45%

3+00.00 107.29 6.45%

3+73.78 112.05 PVC 371.48 6.45%

4+00.00 113.68 6.00%

5+00.00 118.82 4.27%

6+00.00 122.22 2.54%

6+23.78 128.18 PI 350.00 250.00 6.45%

7+00.00 121.26 -6.10%

7+23.78 119.50 PVT 100.00 -8.67%

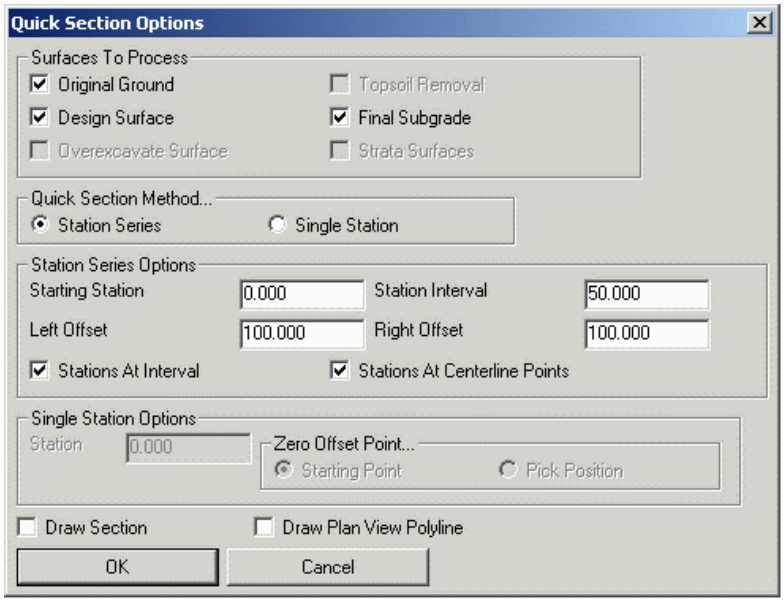
7+75.71 115.00 0.00 51.93 -8.67%

Sample profile report

Quick Section

Function

This command allows you to create, view, edit, and report sections from the TakeOff surfaces.



Surfaces To Process: Here you select which surfaces to include in the section including: Orginal Ground, Design Surface, Overexcavate Surface, Topsoil Removal, Final Subgrade, and Strata Surfaces. If you do not have one of these surfaces in the current project then you can not select them, like Topsoil Removal in the above example.
Quick Section Method...: Station Series will create multiple sections at station intervals along the centerline that you select. Single Station will make one section that follows the selected alignment.
Station Series Options: If you selected station series then here is where you set the starting station and set the interval between each station. The Left and Right Offset is the perpendicular distance away from the station that

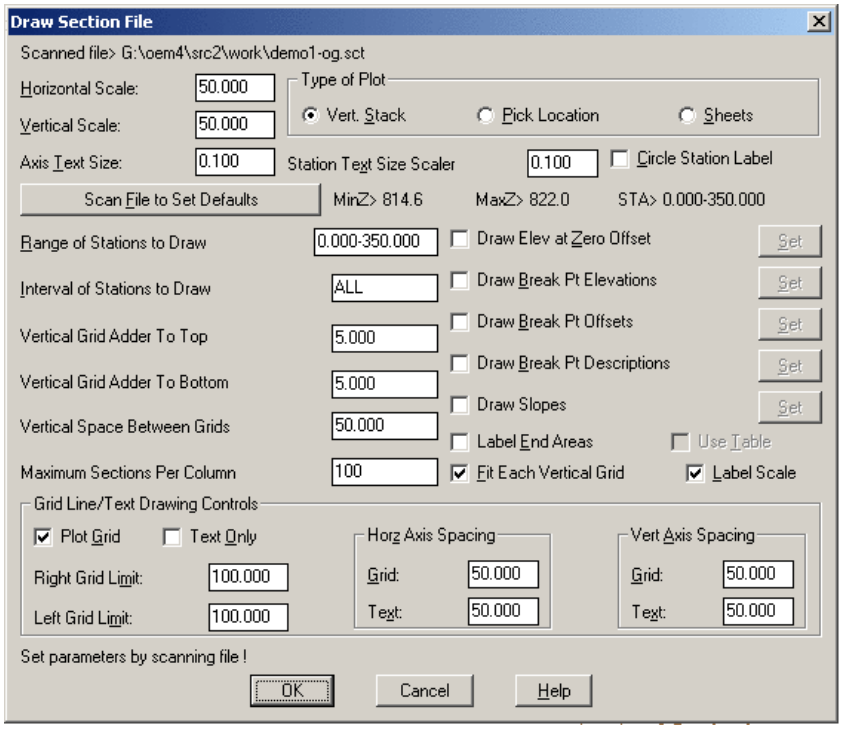
you want to include in the section. It is basically the size of the section. If the Left and Right Offset is 100 feet on each side, then the sections created will be 200 feet across. There is also the option to create stations at each interval or create the stations by points on the centerline. The stations at centerline points is for the PC, PT and ending stations of the centerline.

Single Station Options: If you selected single station then here is where you set the starting station. The Zero Offset Point... determines the base position for the offsets. If your section is 200 feet across and you select the Zero Offset Point Starting Point then the section will be labeled 0 through 200 starting from the first point you made. If you select The Zero Offset Point Pick Position then you will be prompted to pick a zero point. If you pick in the middle of a section 200 feet across, then the middle point will be 0, the right side will end at 100, and the left side at -100.

Draw Section: allows you to draw the section right on the drawing at the end of the command. Draw Plan View Polyline: draws the polyline you used to make the sections right on the drawing as well.

Pick starting point (CL-Centerline,P-Polyline): To select the section alignment, you can pick points on the screen, type in CL in the command prompt, and select a centerline file, or type in P and choosing a polyline from the screen.

Carlson TakeOff creates a section file for each surface you selected. In the Draw Section dialog box you can edit the file names and choose layers to which they will be saved. You can also set the layers for different properties of the section, including: Grid Text Layer, Main Index Grid Line Layer, Intermediate Grid Line Layer, and End Area Layer.

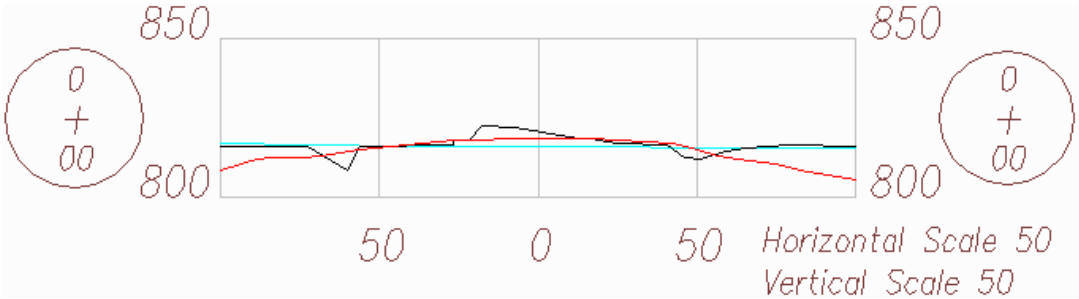


Draw Section File: Here is where you can determine how the section will be drawn.

The Horizontal and Vertical Scale determines the size and the scale of the section drawing. Type of Plot: determines how multiple sections will be arranged, either one on top of the other as a Vert. Stack, by a specified points with Pick Location, or arranged in rows and columns with Sheets.
Axis Text Size and Station Text Scaler determines the size of the text for the section drawing. Circle Station Label: draws circles around station labels for easy viewing.
Scan File to Set Defaults: This routine finds the minimum and maximum elevations and the station range for the section(s) and sets them as the default selections.
Range of Stations to Draw: Here you can specify the range of stations to include in the drawing.
Interval of Stations to Draw: Within that range you can select an interval to only draw a certain among of the stations, or you can type in ALL to draw all the stations within the given range.

Vertical Space Between Grids: is the distance between each grid.
Maximum Sections Per Column: is how many sections are put in each column before rows are created.
Grid Bottom Elevation: is the bottom elevation of the grid.
Vertical Grid Height: is the distance from the bottom elevation of the grid to the top.
The Draw Elev at Zero Offset: labels the elevation at the specified zero offset. Pick on Set to select decimal precision, the text size, the layer, and whether to label on diagonal leaders or not.
Draw Break Pt Elevations: labels the elevations at break points. You can set the decimal precision, the text size scaler, the layer, and which sections to label by on the botton Set.
Draw Break Pt Offsets: labels the offsets of the break points. Pick on Set to select decimal precision, the text size, the layer, and which sections to label.
Draw Break Pt Descriptions: draws the descriptions for the break points. Pick on Set to select decimal precision, the text size, the layer, and which sections to label.
Draw Slopes: Labels the slopes. You can set the text size, layer, format, offset, decimal precision, and which sections to label by picking on the botton Set.
Label End Areas: labels the end areas and allows you to use a table as well.
Fit Each Vertical Grid: Allows for Vertical Grid Adder To Top and Vertical Grid Adder To Bottom. These commands puts spacing between the highest and lowest elevations in the section and the end of the grid.
Label Scale: Displays the scale of the section.

Grid Line/Text Drawing Controls
Plot Grid: Determines whether to show the grid or not.
Text Only: Displays only the text from the grid.
Right Grid Limit, Left Grid Limit: sets the right and left side of the grid.
Horz Axis Spacing Grid Text, Vert Axis Spacing Grid Text: sets the spacing of the grid.
The end result...



Prompts

Command: quicksct

Pick starting point (CL-Centerline,P-Polyline): p

Polyline should have been drawn in direction of increasing stations.
CL File/<Select polyline that represents centerline>:
Loading edges...
Loaded 574 points and 1393 edges
Created 820 triangles

Loading edges...
Loaded 1209 points and 3410 edges
Created 2202 triangles

Loading edges...
Loaded 5057 points and 14923 edges
Created 9866 triangles

Writing section G:\oem4\src2\work\demo1-og.sct
Writing section G:\oem4\src2\work\demo1-bs.sct
Writing section G:\oem4\src2\work\demo1-fn.sct
Initializing Draw Sections command ....
stage3
Text Size in units: 5.0
SN matches
Select Starting Point for Row of Sections <0.0,800.0>:
Scanning File> G:\oem4\src2\work\demo1-og.sct ....
Station> 400.000

Prerequisite: a surface

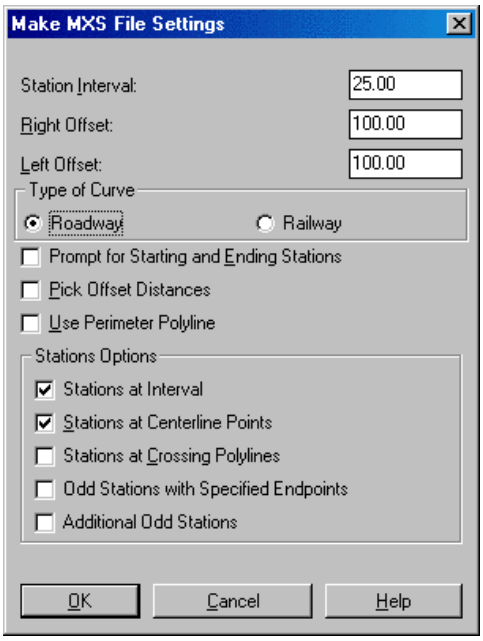
Keyboard Command: quicksct

Input-Edit Section Alignment

Function

This command will create or append to a sectionalignment file which is stored as a Multiple CrossSections (.MXS) file. This file contains the coordinates thatdefine the center and endpoints of section lines and is arequirement of many section commands such as *Sectionsfrom Surface Entities* and *Sections to 3DPolyline*. The section alignment defines the stations along a centerlineand how far left and right to create cross sections.

This routine starts by asking for a new or existingMXS file name. Then the centerline is specified byeither selecting a polyline that represents the centerline orby choosing a centerline file (.CL file). Next theprogram prompts for the starting station of the centerline. Ifthis is a new section alignment, the Make MXS FileSettings dialog appears.



Station Interval: Enter the station interval for sections.

Right Offset: Enter the width for the sections, right of the centerline. Not available if PickOffset Distances is checked.

Left Offset: Enter the width for the sections, left of the centerline. Not available if PickOffset Distances is checked.

Type of Curve: Specify either Roadway or Railroad curve.

Prompt for Starting and Ending Stations:

Pick Offset Distances: Allows you to specify the offsets by using the distance between twopicked points in the drawing.

Use Perimeter Polyline: Allows you to specify a closed polyline that will be used as the limitof the cross sections. The offsets will be contained within this closed polyline.

Station Options: There are five methods for locating the stations:

Stations at Interval: Creates cross sections at the specified interval such as every 25 feet. Ifthe Prompt for Starting and Ending Stations is on, then the program will apply the station intervalto the user-specified range of stations. Otherwise the station interval is used along theentire centerline.

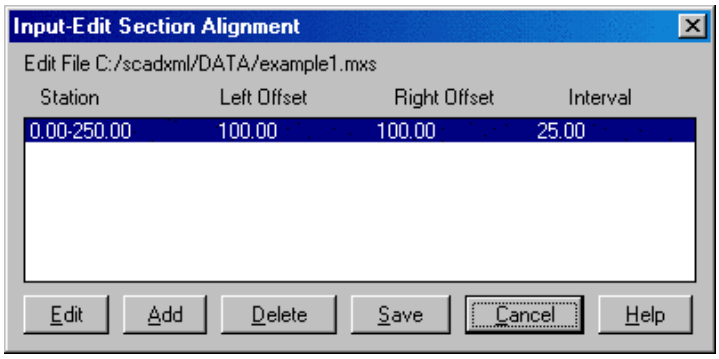
Stations at Centerline Points: Creates cross sections at every transition point in thecenterline such as the PC, PT, spiral points and end points.

Stations at Crossing Polylines: Allows you to select polylines that cross the centerline andcreates cross sections at the intersections of these polylines with the centerline.

Odd Stations with Specified Endpoints: Creates cross sections at stations that are entered orat picked points along the centerline. This option also allows you to pick the left and right offsetpoints which do not have to be perpendicular to the centerline.

Additional Odd Stations: Creates cross sections at the specified stations but the offsets arealways perpendicular to the centerline with the user-defined default offset distances.

The Input-Edit Section Alignment dialog lists all the section stations and offsets in the alignment.



Edit: Allows you to edit the currently highlighted row

Add: Allows you to add more sections.

Delete: Deletes the currently highlighted row.

Save: Saves the MXS file, exits this dialog and draws the section alignment on the screenusing temporary vectors (yellow for left offsets, magenta for right offsets). Any viewport changesuch as *Redraw* or *Zoom* will cause these vectors to dissappear. The draw the section lines withLine entities, use the *Draw SectionAlignment* command.

Prerequisite: A polyline centerline or centerline file

Keyboard Command: editmxs

Sections From Existing Surface

Function

This command will create a section file (.sct) for the existing surface. First, select a .mxs file for section alignment. If you need to create a .mxs file go to Input_Edit Section Alignment. Next, save the file with a new name or override a pervious file and the section file will be created. You can now use the other Section commands to draw, edit, and report from this section.

Prompts

Command: sctgrid2
Reading edges 1393, intersections found 541

Prerequisite: a surface

Keyboard Command: sctgrid2

Sections From Design Surface

Function

This command will create a section file (.sct) for the design surface. First, select a .mxs file for section alignment. If you need to create a .mxs file go to Input_Edit Section Alignment. Next, save the file with a new name or override a pervious file and the section file will be created.You can now use the other Section commands to draw, edit, and report from this section.

Prompts

Command: sctgrid3
Reading edges 1393, intersections found 541

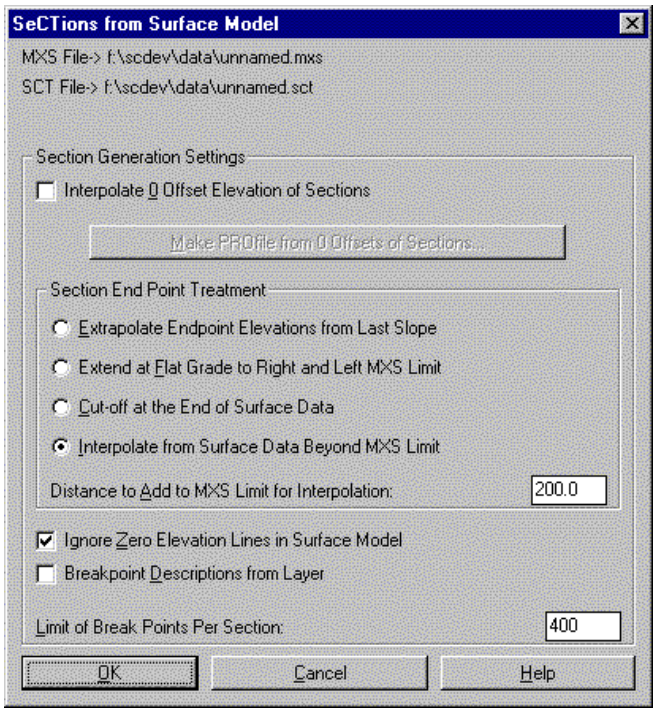
Prerequisite: a surface

Keyboard Command: sctgrid3

Sections from Screen Entities

Function

This command allows you to create cross sections from a surface model. The stations for the sections and the left and right offset distances are defined in the MXS file that must be created before running this routine using Input-Edit Section Alignment. The surface model is defined by lines or polylines with elevation. The polylines with elevation could be a contour drawing file from a photogrammetry firm or can be created from survey points with the *Triangulate & Contour* command. When using Triangulate & Contour, it is useful to use the Draw Triangulation Lines option because the 3D triangulation lines represent all the break lines in the surface which increases the accuarcy of the cross section, as against just using the contours. Barrier lines or 3D polylines can also be used to represent break lines along ridges and valleys. The program samples the selected lines, polylines and 3DFace entities and calculates the intersections of these segments with any of the cross sections. The station, offset and elevation of these intersections make up the data in the section file. This section file (.SCT file extension) can be reviewed or edited with the


SeCTions from Surface Model
MXS File-> f:\scdev\data\unnamed.mxs
SCT File-> f:\scdev\data\unnamed.sct
Section Generation Settings
Interpolate 0 Offset Elevation of Sections
Make PROfile from 0 Offsets of Sections...
Section End Point Treatment
Extrapolate Endpoint Elevations from Last Slope
Extend at Flat Grade to Right and Left MXS Limit
Cut-off at the End of Surface Data
Interpolate from Surface Data Beyond MXS Limit
Distance to Add to MXS Limit for Interpolation:
200.0
Ignore Zero Elevation Lines in Surface Model
Breakpoint Descriptions from Layer
Limit of Break Points Per Section:
400
OK
Cancel
Help

Input-Edit SeCTion File command. Also the section file can be plotted with the *Draw Section File* command or used in the by the *Earthworks and Final Contours* command to calculate volumes.

The options for this command are set in the dialog shown below. The Interpolate 0 Offset Elevation of Sections option will add a data point at offset zero for every station with an elevation that is interpolated from existing offsets. The Breakpoint Descriptions from Layer option will store the layer name of the surface entity as the description for the offset-elevation point in the section file. The section end points are the left and right furthest offsets such as left and right 100 feet. When calculating sections based on the intersections with surface entities, there usually, intersection exactly at the end points is not possible. For example, there could be contours at offsets right 87.31 and 105.43 but no intersection exactly at 100. There are four methods for determining the elevation for these end points. (1) The Extrapolate Endpoint Elevation from Last Slope calculates the slope from the last two offset-elevation points and calculates the elevation at the endpoint from this slope. For example, given offsets at 80 with elevation 100 and 90 with elevation 101, the elevation at offset 100 would be 102. (2) Extend at Flat Grade to Right and Left MXS Limit uses the last offset elevation as the end point elevation. For example, if the last offset were 85 with elevation 102, the program would add an offset at 100 with elevation 102. (3) The Cut-off at the End of Surface Data option does not add offsets at the end points. The sections will end at the last offset found in the surface model. (4)Interpolate from Surface Data Beyond MXS Limit looks beyond the offset limit for more intersections with surface entities. Then the endpoint elevation is interpolated between the offsets above and below the endpoint. For example, given offsets at 90 with elevation 101 and at 110 with elevation 103, the endpoint offset at 100 would have elevation 102.

Prompts

MXS File to Process Select the section alignment file

New or Append Choose whether to create a new section file or add to an existing section file

Section File to Create Specify the section file

Select Lines, PLines, and/or 3DFaces that define the surface.

Select objects: *pick the surface entities*

Compiling file c:\scdev\data\simo2.sct

Prerequisite: Construct surface model to be sampled

Sections from Grid or TIN File

Function

This command creates a cross-section file (.SCT file) from a surface model that is defined by a 3D rectangular grid file (.GRD file) or a triangulation file (.TIN). The grid file can be created in the DTM-Contour module with the *Make 3D Grid File* routine. The triangulation file can be created with the Write Triangulation File option in the *Triangulate & Contour* command. This command also requires an .MXS file to define the alignment and stations of the sections. The number of section points created is displayed at the end of the routine.

Prompts

Choose Grid file to process

Choose MXS File to Process

Choose SCT file to write

Found 1410 section points.

Prerequisite: A grid file (.GRD file) or triangulation file (.TIN file) and a cross sections alignment file (.MXS file)

Keyboard Command: profedit

Sections from Polylines

Function

This command allows the user to select a polyline that represents a section in cross section view and writes it to a .SCT file. This can be useful for revising sections or for defining a new one. For example, let's say you have extracted sections from a surface model of the existing ground on a site and have plotted them using the *Draw Section File* command. Now use this command to send the sections to a Section file and compute the earthworks using the *Calculate Sections Volume* command. After selecting the command, the Polyline to Section File dialog appears.

The first time this command is selected the output Section file is set to the same name as the current drawing. Select the Section File Name... button to specify a different name.

The Station Interval edit box allows you to specify the amount that the default station number will be incremented as the station prompt shown below appears.

The Interpolate Zero Offset toggle if on, causes the program to output the elevation of the zero offset to the output .SCT file.

A second section file can be specified to process two sections at a time for each station. This allows you to handle both existing and final grades at once.

After selecting the OK button the prompts below appear.

Prompts

Command:
SCTFPL
Exit/Pick text/<Station <0.0000>>: Press Enter for the default
Exit/Pick text/<Starting elevation of grid <100.0000>>: Pick Text from the screen

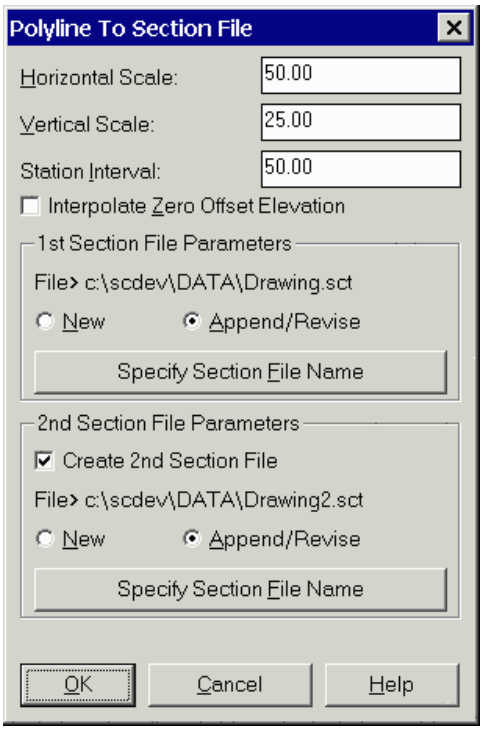
[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): Press Enter
Select station 0.0000 section polyline: Pick a Polyline
Renamed original file> C:/TAKEOFF_2004/DEMO2A-OG.SCT as> C:/TAKEOFF_2004/DEMO2A-OG.sck
1 -1766.0303 832.1150
2 -1768.3750 805.5000
3 -1765.7500 780.0000
List continues...
Sta> 0.000 Revised data stored in C:/TAKEOFF_2004/DEMO2A-OG.SCT
Exit/Pick text/<Station <50.0000>>: Press Enter for the default
Exit/Pick text/<Starting elevation of grid <100.0000>>: Pick Text from the screen

[int on] Pick point at starting elevation and zero offset of section ([Enter] for none): Press Enter

Select station 50.0000 section polyline: Pick a Polyline

Renamed original file> C:/TAKEOFF_2004/DEMO2A-OG.SCT as> C:/TAKEOFF_2004/DEMO2A-OG.sck
1 1857475.2197 159052.3650
2 1857472.8750 159025.7500
3 1857475.5000 159000.2500
List contiunes...
Sta> 50.000 Revised data stored in C:/TAKEOFF_2004/DEMO2A-OG.SCT
Exit/Pick text/<Station <50.0000>>: Exit
Prerequisite: Plot the section or profile to write to the .SCT file.

Keyboard Command: sctfpl



Sections from Points

Function

This command creates an .SCT file from Carlson points in the drawing. An .MXS file is needed to define the centerline and the stations of the cross sections. The offsets for the cross sections points are derived from the perpendicular distance between the centerline and the TakeOff points. The cross section elevations come directly from the elevations of the points. In order to be included in a cross section, a TakeOff point must be within the offset tolerance distance of the cross section line.

Prompts

Choose MXS File to Process

Choose SCT file to write

Enter the maximum offset tolerance <1.0: *Press Enter*

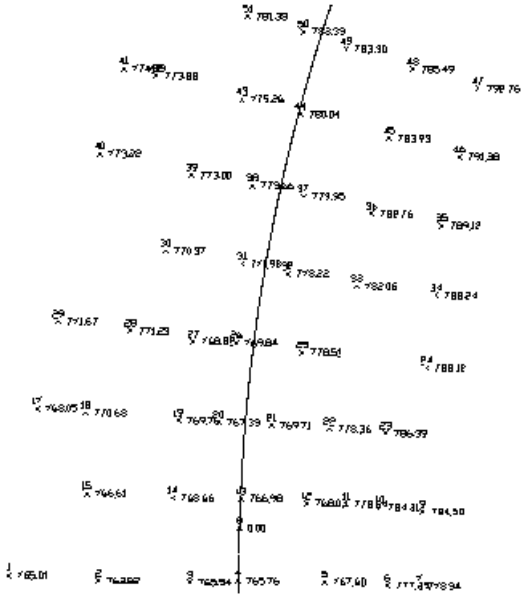
Ignore Zero Elevations (<Yes>/No)? *Press Enter*. This option will filter out all Carlson points that have a zero elevation.

Select the Carlson points along the sections.

Select objects: *pick the Carlson point inserts*

Prerequisite: Carlson points and an MXS file

Keyboard Command: sctpts



TakeOff points for use in creating Section file

Import Sections

Function

This command converts Columnar Text, Agtek, Ceal, GeoPak, IGRDS, Moss, RoadCalc, SMI, and Softdesk files into Carlson TakeOff section (.section) files.

Prerequisite: a Columnar Text, Agtek, Ceal, GeoPak, IGRDS, Moss, RoadCalc, SMI, or Softdesk file

Sections to 3D Polylines

Function

This command creates 3D polylines from a section file (.sct). Besides the section file, a centerline polyline or section alignment file (.MXS) must be specified to define the plan view location of the 3D polylines. The elevations for the 3D polylines come from the section file. These 3D polylines can then be used by other Carlson routines to create surface models.

Typically the 3D polylines are drawn as cross-sections perpendicular to the centerline at each station. When using a polyline centerline instead of the MXS file, there is an option to draw by connecting similar descriptions to make 3D polylines parallel to the centerline. For example, if the section file has descriptions for each section point then you can draw 3D polylines for EP, SHD, TIE, etc.

Prompts

Layer Name for 3D Polylines <3DXSEC>: *Press Enter*

Align sections by MXS file or polyline centerline (MXS/<Polyline>)? *Press Enter*

Choose Section File to Process Select the .sct file

Range of stations: 1.14 to 1605.25

Enter the starting station to process <1.14>: *Press Enter*

Enter the ending station to process <1605.25>: *Press Enter*

Draw sections or offset polylines by description (<Section>/Offset)? *Press Enter*

Select centerline polyline: *pick the polyline*

Enter the centerline starting station <0.0>: *Press Enter*

Draw perimeter of sections (Yes/<No>)? *Y for yes.* This option will connect all the left most offsets and right most offsets together with a 3D polyline.

Prerequisite: A section file (.sct)

Keyboard Command: scto3dp

Sections to Points

Function

This command creates Carlson points using a section file (.SCT file) to define the point elevations. The x,y position of the points are calculated based on the station and offset along a centerline polyline. These points are stored in the current .CRD coordinate file and can also be plotted in the drawing. Points can be created at each station in the section file or at a set station interval. The range of stations to process can also be set. The Description Match field can be used to filter the offsets and only create points with matching descriptions (ie only "EOP" offsets). The *Create points at fixed offsets* option can be used to make points at user-specified offset distances. The program will interpolate the elevations for these points by interpolating from the neighboring offsets. The Reduce Points option will skip creating points for the same offset between stations if the x,y position and elevation change is less than the offset tolerance. Essentially, when a series of offsets are on a straight line (no vertical and no horizontal curve) then only the starting and ending points are needed and all the intermediate points can be skipped. For example, the Reduce Points routine will look at the left side "EOP" offset points at stations 1+00, 1+05 and 1+10 and if these three points make a straight line then the point for station 1+05 can be reduced. The Offset Distance is the tolerance that Reduce Points using for testing whether the middle point (offset point at station 1+05) can be reduced. The distance for the middle point is calculated as the perpendicular distance from the middle point to the line between the two end points. Both the horizontal and vertical distances are checked.

Prompts

Sections to Points Settings dialog

CooRDinate File to Process

Choose a .CRD file to add the points to. This prompt only occurs if no CRD file is current.

Choose SCT file to read Pick the cross section file.

Range of stations: 3.34 to 750.00

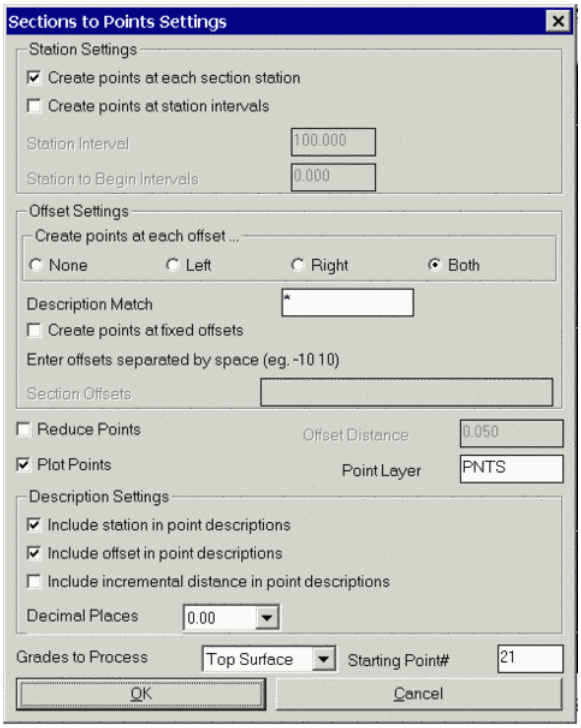
Enter the starting station to process <3.34>: *Press Enter*

Enter the ending station to process <750.00>: *Press Enter*

Select centerline polyline: *pick the polyline that defines the stations*

Enter the centerline starting station <0.0>: *Press Enter*

Prerequisite: .sct file & polyline centerline
Keyboard Command: sctopt



Slope Zone Section Analysis

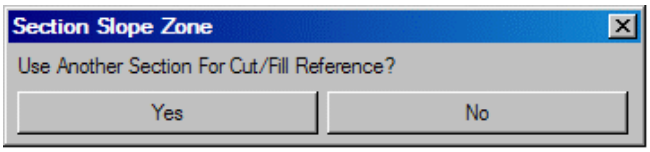
Function

This command reports the cut/fill areas and volumes within given ranges of slopes. There is an option to use another section for cut/fill reference.

Prompts

Select Section to Process ***Select .SCT file***
Select Slope Zone dialog ***Select No***
Report slope or horizontal area [<Horizontal>/Slope]? *s*
Slope format [<Percent>/Ratio]? *Enter*
Greatest slope % of zone 1: ***3***
Greatest slope % of zone 2: *Enter*

Starting station to process <0.000>:
Ending station to process <0.000>: 1000
The Standard Report Viewer creates a report called Section Slope Zone Analysis Report.



Keyboard Command: sctzone
Prerequisite: .SCT file

Highway Section Staging

Function

This command applies to the situation of designing a partly completed road or regrade. Using an existing and a final grade section file, the program will create four new sections files for the finished existing sections, finished final sections, remaining existing sections, and remaining final sections. The source existing and final section files should have matching stations. There is an option to process a range of the possible stations from the section files. The complete part of the road can be either on the left or right side. The pivot point is cross section offset where the completed part ends. From this point, the final grade will connect to the existing grade by a line at the specified slope.

Prompts

Select Existing Sections File

Choose the cross sections file.

Select Final Sections File

Choose the cross sections file.

Enter slope as percent grade or slope ratio (Percent/<Ratio>)? *Press Enter*

Enter the fill slope ratio <2.0>: *Press Enter*

Enter the cut slope ratio <2.0>: *Press Enter*

Place road on left or right (<Left>/Right)? *Press Enter*

Range of stations: 50.0 to 100.0

Enter the starting station to process <50.0>: *Press Enter*

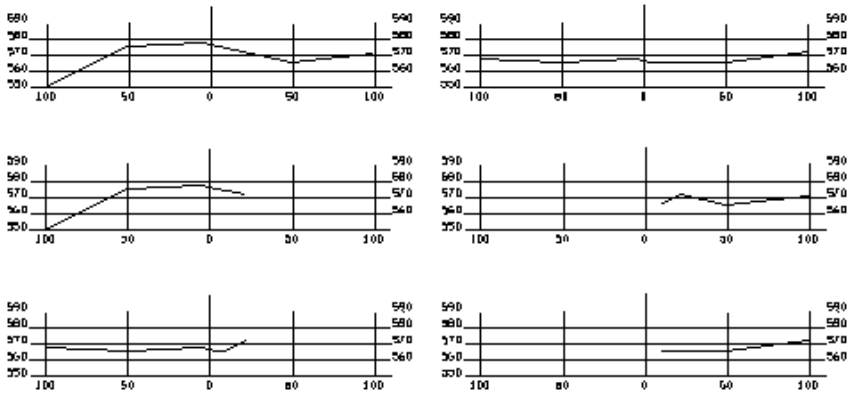
Enter the ending station to process <100.0>: *Press Enter*

Apply same pivot offset to all stations (Yes/<No>)? *Yes*

Enter the pivot offset (enter left offsets as negative) <0.0: *5.0*

Prerequisite: Existing and final grade section files (.SCT)

<>**Keyboard Command:** sctstage



</>

Input-Edit Section File

Function

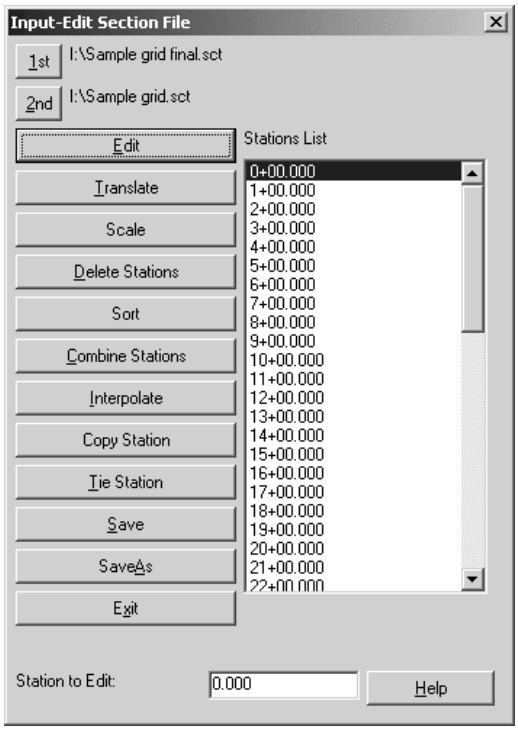
This program can be used to enter or edit data stored in a section file (.SCT file), including a real-time graphic window in the Edit mode. The section data consists of stations, offsets, elevations and descriptions. This command also has utilities for translating the offsets and elevations, deleting stations from the file, intersecting the outslopes of one section file with another, combining multiple occurrences of the same station and sorting the stations, offsets and elevations. While editing the section file, a second section file can be used as reference. To choose this file, pick the 2nd button. For example when editing the proposed section file, you can view the ground section file.

The program begins by prompting for a New or Existing section (.SCT) file. Use the new option to create a new file. Use the existing option to edit the offsets and elevations for station/sections that you have already created or append new stations to a file.

The standard file dialog appears allowing you to specify the file that you want to operate on. The program defaults to a section file the same name as the drawing or a name that you specified using another section command.

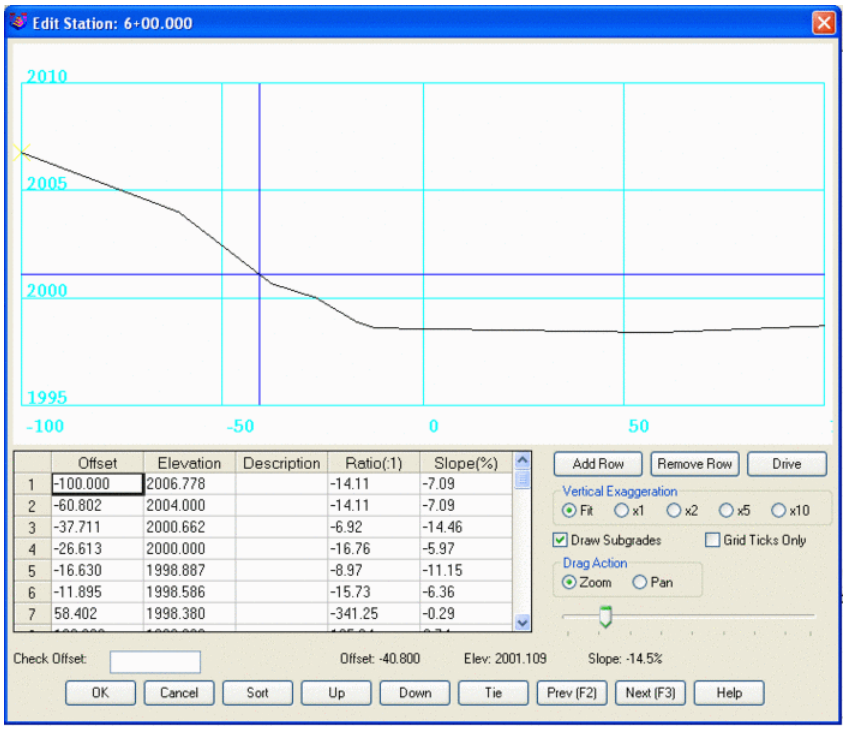
After specifying the file name the program displays any stations currently in the file in the Stations List.

If you specified a new file the Stations List box will be blank. To edit and display the offset and elevation data at a station you can double click on the station in the list box or input the station in the Station to Edit edit box at the bottom of the dialog. To add a station to a new file or existing file you must enter the station in the Station to Edit edit box. After selecting a station number the program will display a dialog arranged in two columns, one of the left and one of the right offsets and elevations.

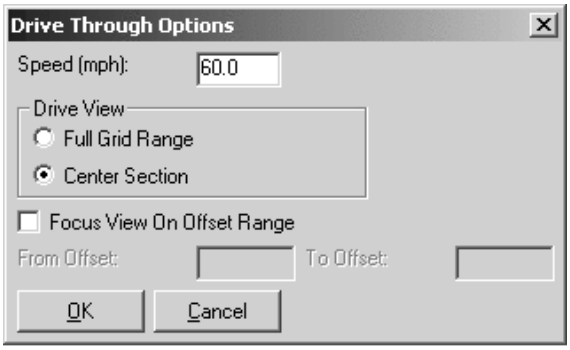


Edit: Opens the Edit Station dialog which shows a graphic of the section on top, a list of the offset-elevation points in the middle, and the function buttons on the bottom. To add an offset point, type in the offset, elevation and description in the spreadsheet. Left offsets are entered as negative numbers. To edit an offset point, highlight the point from the list and then edit the values in the Offset, Elev and Desc columns. The Sort button will sort the list of offsets from lowest to highest, left to right. The Up button will move the highlighted offset point up in the list. Likewise the Down button moves the highlighted offset point down in the list. The Delete button erases the highlighted offset and elevation from the list. After inputting or editing press the OK button to return to the Stations List dialog and keep any changes you have made. Select the Cancel button if you want to cancel changes made to the current station.

A real-time report of offset-elevation-slope now displays as you move the cursor across the section in the graphic window. A Drive function scrolls through the sections at a user-specified speed. Edit subgrades has the ability to tie a subgrade into the surface. A new Check Offset field calculates an elevation based on an entered offset. In the elevation edit field, equations (+,-,*,/) can be entered to calculate or adjust an elevation.



Drive: This function scrolls through the sections at the rate of speed specified by the user in the Speed window. The Drive View options determine whether the sections are displayed using the full width of the graphic window or centered in the window. Focus View On Offset Range allows the user to set the left and right viewing limits of the sections. Section data beyond the specified limits is not displayed.

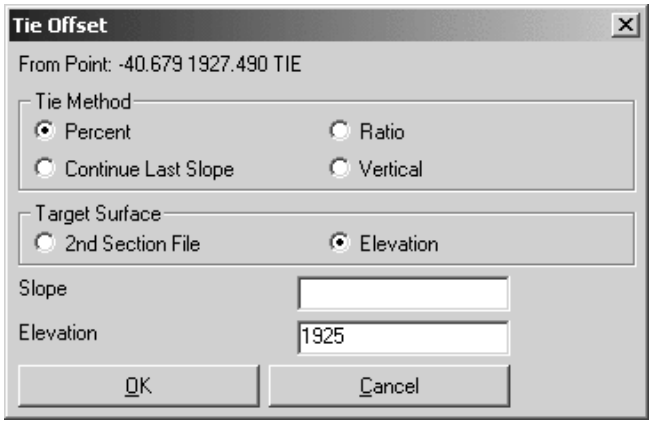


Drag Action: Determines whether holding down the right mouse button functions as "Zoom" or "Pan" in the viewer window. Holding down the scroll button also acts as a Pan function.

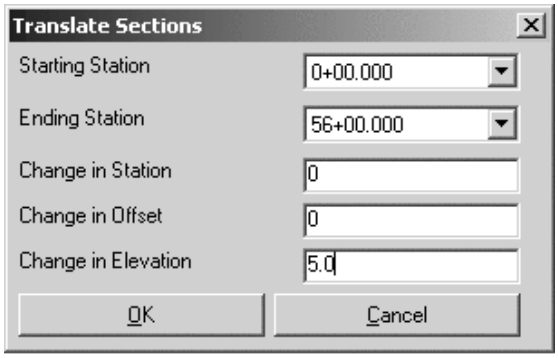
Check Offset: Calculates the elevation for the offset value entered in the box. Use the negative sign (-) for left offsets.

Elevation Field: Equations can be used in this field to calculate or adjust an elevation. For instance, to subtract 1.25' from elevation 1926.18, simply enter 1926.18-1.25 and press enter. The new elevation will be calculated and displayed in the viewer window. Functions for add, subtract, multiply and divide are supported.

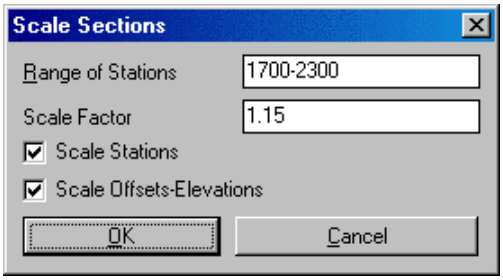
Tie: The Tie dialog layout with a new option to tie the section to a specified elevation in addition to a surface (second section file).



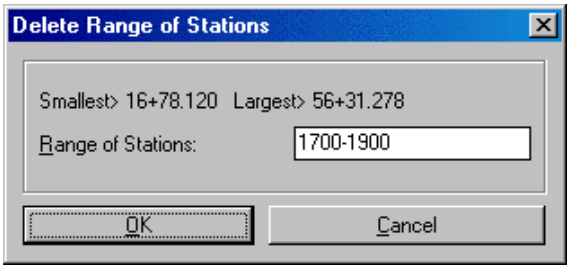
Translate: Allows you to add or subtract a distance from the offsets to adjust or shift the centerline. You can also adjust the elevations up or down. When using this option you are prompted for the range of stations to operate on and the values to adjust the offsets and elevations. If for example you want to shift the centerline but not the elevations, enter the plus or minus amount you want to translate and when prompted for the elevation enter zero.



Scale: Allows you to scale the station, offsets and/or elevations by the specified scale factor. This function can be used to convert between English and metric units.



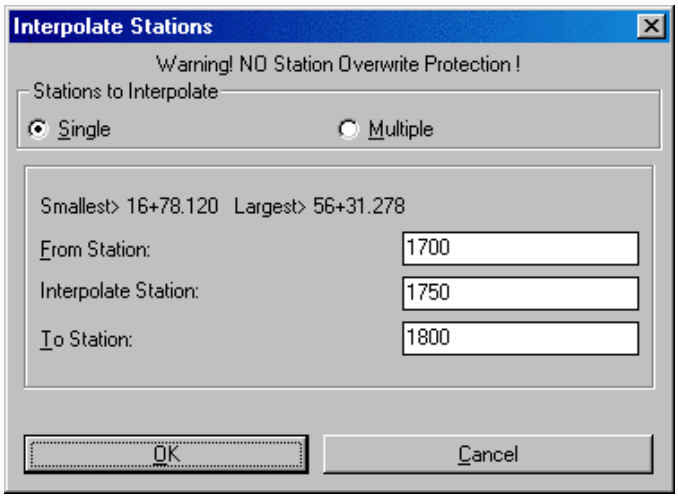
Delete Stations: Allows you to remove a station or range of stations from the in memory Stations List. When prompted by the dialog above enter a single station or enter a range by entering the smallest or lowest station number separated by a dash or any non numeric character, and the largest or highest station number. After specifying the range select the OK button to proceed with the removal or select the Cancel button to abort the deletion process. Since the station editor data is stored in virtual memory, if you accidentally delete a range, Quit the editor with out saving the stations to disk. Then recall the original file.



Sort: Allows you to sort the station numbers into ascending order and sort the offsets and elevations in the individual station records (offsets are sorted from left to right). When sections are derived from the *Sections from Surface Model* command they are already sorted, but when sections are digitized or input manually they occur in the order that you digitized them, so for proper plotting and earthworks you may want to run the sort option before processing.

Combine: Used to bring together in one record slot multiple occurrences of the same station number. This can occur when using the Digitize Sections (XSec) command and the section that you are digitizing has match/break lines which forces you to digitize the station in two or more parts.

Interpolate: Allows you to add or overwrite a station between two stations or projecting forward from two stations. When the dialog appears specify the two known stations in the From and To Station edit boxes and the new station to interpolate in the Interpolate Station edit box. Select the OK button to execute the function with the current settings or select the Cancel button to abort the process.



Copy Station: Allows you to copy a station that already exists to a new or existing station number. Specify the existing From Station in the edit box then the new station number in the To Station edit box. Select the OK button to execute the function with the current settings or select the Cancel button to abort the process.

Tie Station: Allows you to tie the outslopes into the reference second section file. This routine first brings up a dialog to specify the range of stations to process. Next there is a dialog to set the slope to tie with. The program will start from the left most offset and use this slope to find the intersection with the reference section file. Then the intersection from the right most offset is calculated with this slope. These intersection points are the tie points. The slope can be defined by ratio, percent, width/offset, horizontal, vertical and to continue the last slope.

Save: Saves the currently loaded section file.

SaveAs: Allows you to save the currently loaded section file as a different file.

Quit: Allows you to exit from the section editor and return to the drawing editor. The program will warn you to save to a file if you have made changes.

Prerequisite: None

Keyboard Command: scted

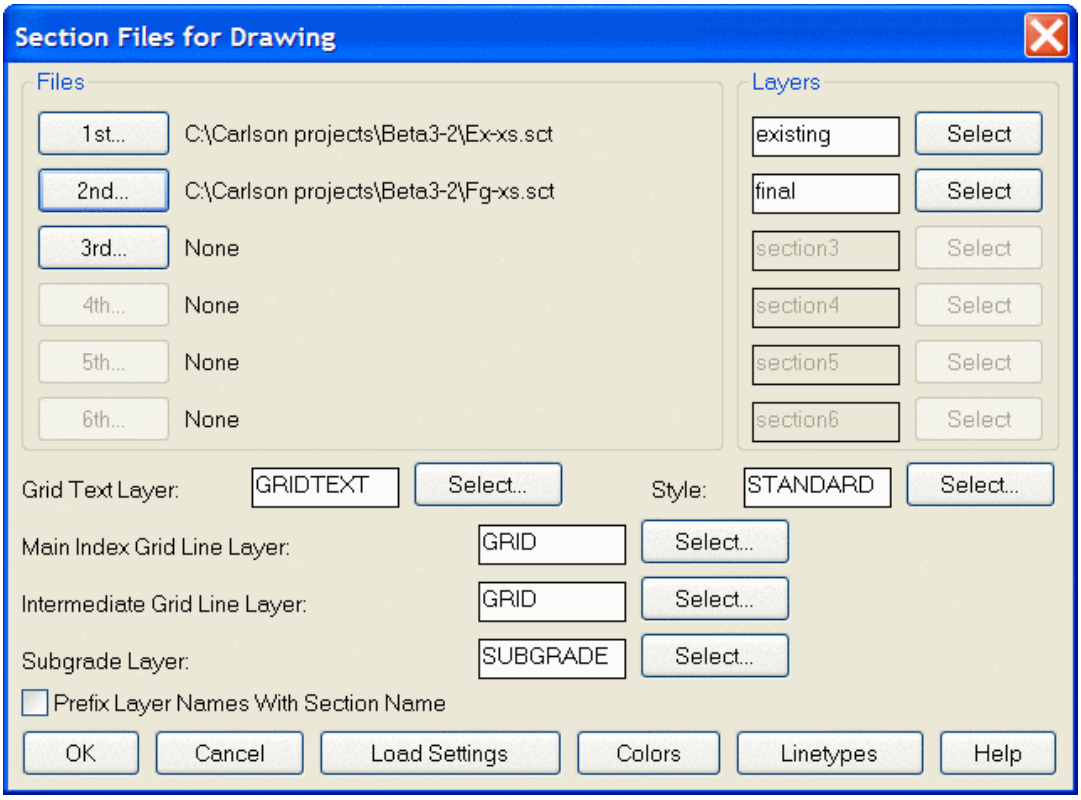
Draw Section File

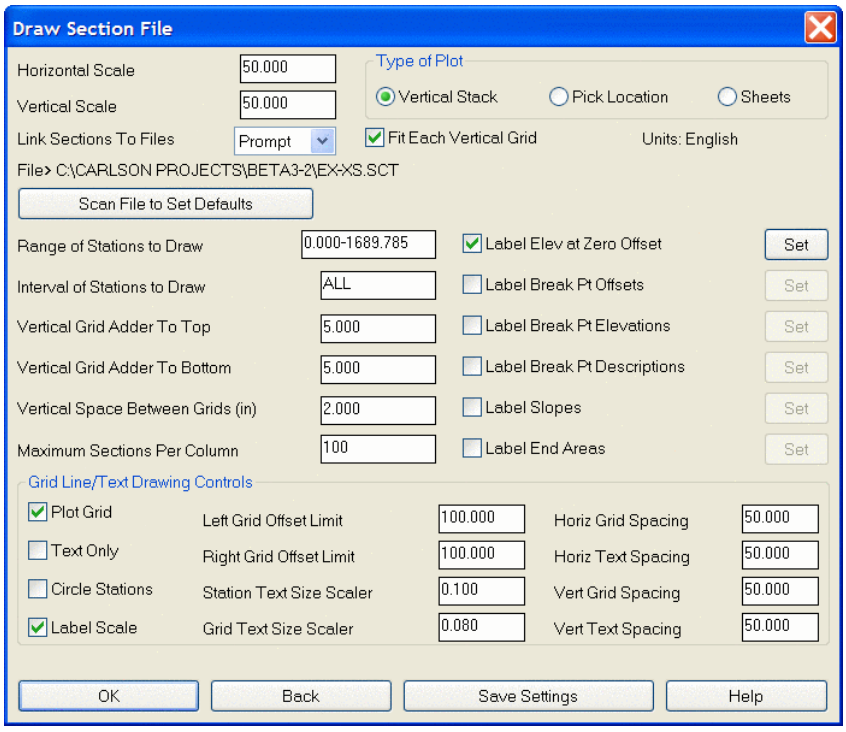
Function

This command will plot the section data from up to six section (.SCT) files at once. The section file can be created by several methods including *Input-Edit Section File, Sections from Surface Entities, Digitize Sections* or *Process Road Design* command. A range of sections can be plotted in a vertical stack, on section sheets, or by selecting a point that corresponds to the grid bottom elevation.

When drawing sheets format in metric mode, be sure to set metric On (clicked) in the Drawing Setup command. Then in the Sheet Parameters dialog, set the Block Name to SCTSHT2 and set your metric sizes.

In the initial dialog, specify up to six section (.SCT) files to plot, the layer for each, and the layer names and text styles for the overall grid text, grid lines, and subgrade. There is also a toggle to Prefix Layer Names with Section Name, so that all layers created for the sections begin with the section name. At the bottom of the dialog is a button to Load Settings, loading a set of previously saved settings, and buttons to set the Colors and Linetypes for the section components.





The second dialog box presents the next level of settings for the generation of sections.

Horizontal Scale: Specify the horizontal scale.
Vertical Scale: Specify the vertical scale.
Link Sections To Files: This setting controls the linkage of the plotted sections to the actual section file(s) (.SCT), determining how changes to the file affect the plotted sections. If set to Off, there is no linkage, Prompt will ask whether to update the plotted sections when the file changes, and Auto will automatically update the plotted sections when the file changes.
Type of Plot: Specify how the sections will be plotted, either as a Vertical Stack, Pick Location, selecting the datum point of each section, or Sheets, which will plot the sections on a block section sheet.
Fit Each Vertical Grid: When checked, the grid bottom elevation and grid height are set automatically, and you may specify values to add to the top and bottom of each grid. See Vertical Grid Adder to Top and Vertical Grid Adder to Bottom. When not checked, the Vertical Grid Adder to Top and Vertical Grid Adder to Bottom options change to Grid Bottom Elevation and Grid Vertical Height.
Scan File to Set Defaults: This button allows the program to set the minimum and maximum parameters. If you choose this option the program will automatically set the range of stations, vertical spacing distance, right and left grid distances and starting/datum elevation. This option writes a file called "sectsort.tmp" that is read and used to set the defaults the next time you use the program. Therefore, if you are selecting a different .SCT file to plot you should use this option to update the .TMP file.
Range of Stations to Draw: Specify the range of stations from the file which will be drawn.
Interval of Stations to Draw: Specify the interval of stations to draw. For example, perhaps you sampled every 25 feet with the *Sections from Surface Model* command for more accurate quantities but only want to plot 50 foot stations. ALL is the default value for this field.
Vertical Grid Adder to Top: Specify the distance that will be added to the highest elevation of the section for the sheets and pick location options. Only available when Fit Each Vertical Grid is checked ON.
Vertical Grid Adder to Bottom: Specify the distance that will be subtracted from the lowest elevation of the section for the sheets and pick location options. Only available when Fit Each Vertical Grid is checked ON.
Grid Bottom Elevation: Specify actual bottom elevation for each section grid. Only available when Fit Each

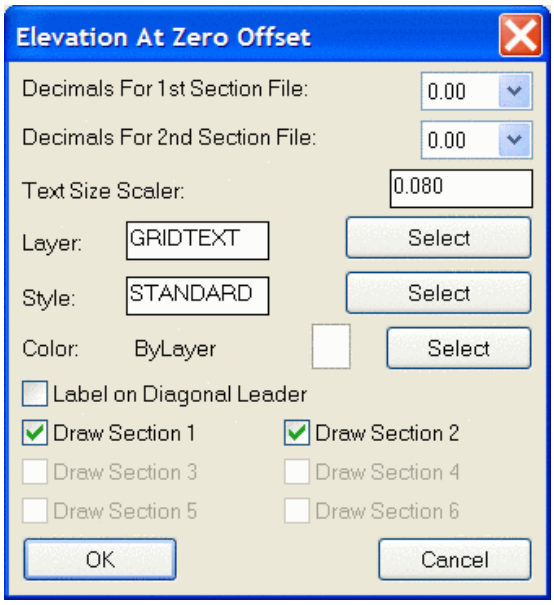
Vertical Grid is checked OFF.

Vertical Grid Height: Specify actual grid height for each section grid. Only available when Fit Each Vertical Grid is checked OFF.

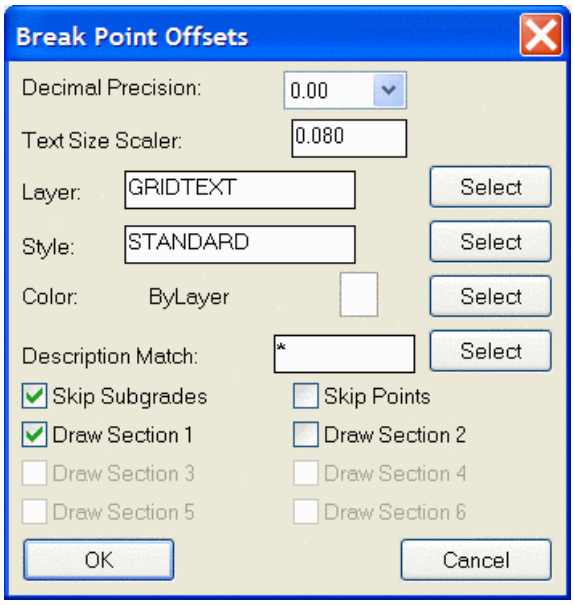
Vertical Space Between Grids: Specify the distance the sections are stacked above the last one plotted when drawing multiple sections.

Maximum Sections Per Column: Sets the maximum number of sections allowed per column.

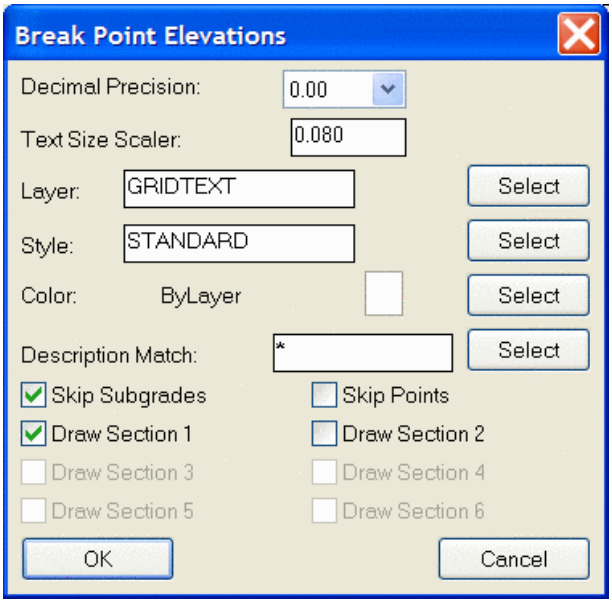
Label Elevation at Zero Offset: Will label the section elevation at offset zero. The label is drawn on the section grid just above the section line. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels. There is also an option to draw the elevation on a 45 degree diagonal, otherwise the elevation label is draw vertically.



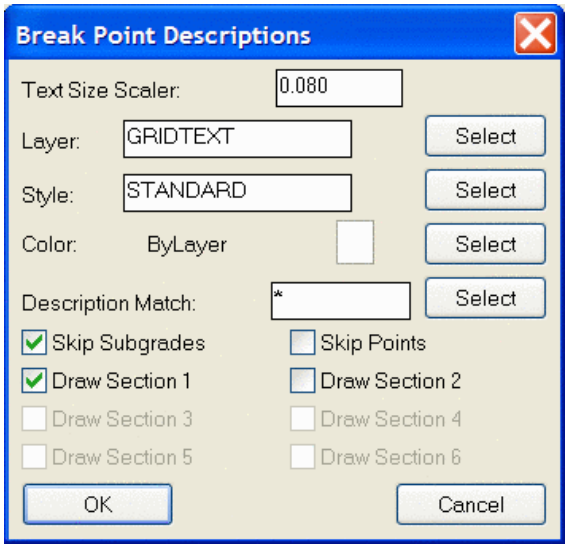
Label Break Pt Offsets: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels.



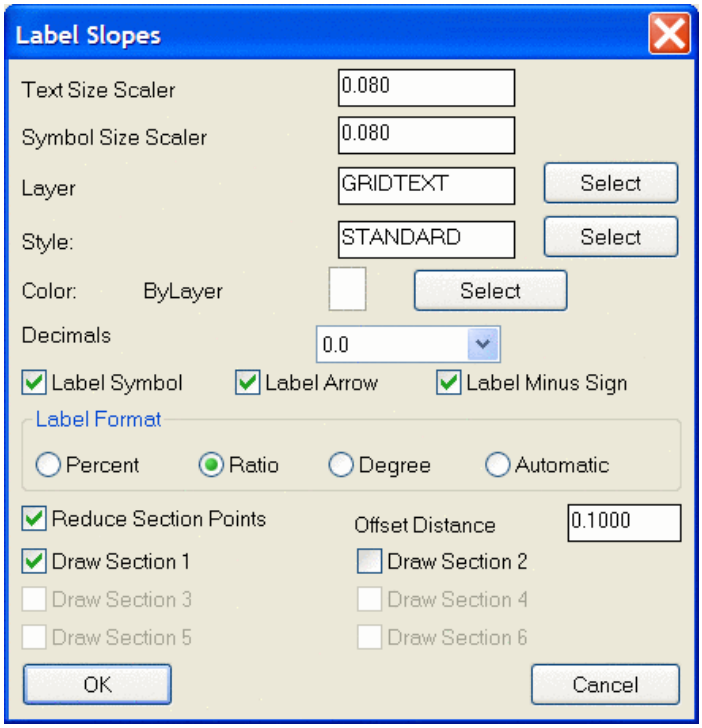
Label Break Pt Elevations: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the display precision, text size scaler and layer for these labels.



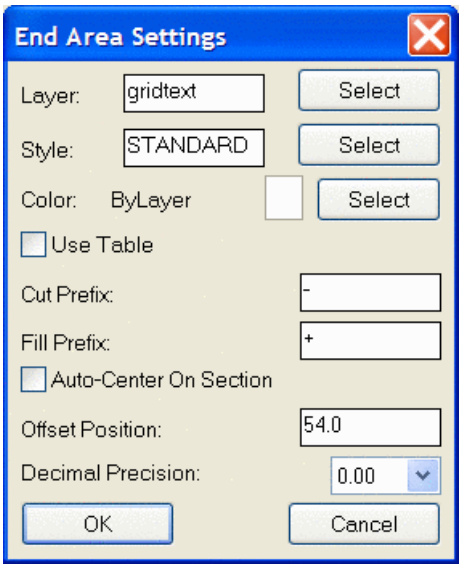
Label Break Pt Descriptions: Will label these values along the section line above each point in the section. Press the Set button to the right of this toggle to set the text size scaler, layer, and description match for these labels.



Label Slopes: Will draw in the slopes.



Label End Areas: Will label cut and fill quantities on each section.
Use Table: Puts end areas cut/fill values in table.



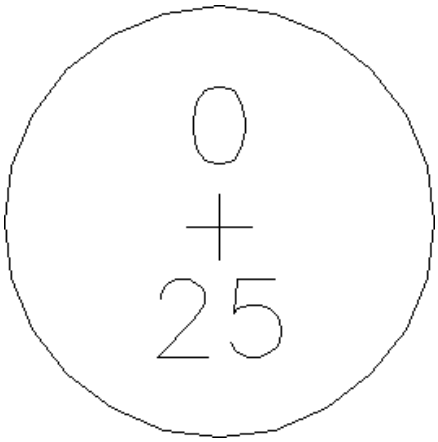
Grid Line/Text Drawing Controls

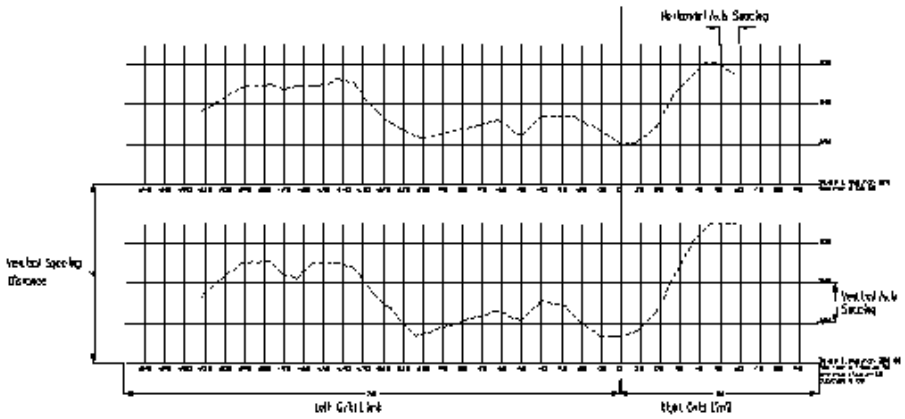
Plot Grid: Uncheck this toggle if you do not want the grid to plot.
Text Only: Check this toggle if you only want to plot the cross section polyline and the grid text. This can be useful for plotting on a section sheet that has pre-plotted grid lines and you want to plot only the section and text.
Circle Stations: Will draw the station number with a circle around it on the left and right sides of the section grid.
Label Scale: Will label the horizontal and vertical scale on each section.
Left Grid Offset Limit: Specify the length the grid lines are plotted to the left from the centerline or zero offset.
Right Grid Offset Limit: Specify the length the grid lines are plotted to the right from the centerline or zero offset.
Station Text Size Scaler: Specify the text size scaler for the station text. This value is multiplied by the horizontal scale to obtain the final text height. For example, if you set Station Text Size to 0.10 and the horizontal scale is 100.0, then the text height will be (0.10 X 100) or 10.0.
Grid Text Size Scaler: Specify the text size scaler for the axis text. This value is multiplied by the horizontal scale to obtain the final text height. For example, if you set Axis Text Size to 0.08 and the horizontal scale is 50.0, then the text height will be (0.08 X 50) or 4.0.
Horiz Grid Spacing: Specify the distance the vertical lines of the grid will be spaced.
Horiz Text Spacing: Specify the interval that text will be plotted below the grid lines.
Vert Grid Spacing: Specify the distance the horizontal lines of the grid will be spaced.
Vert Text Spacing: Specify the interval that text will be plotted to the left and right of the grid lines.

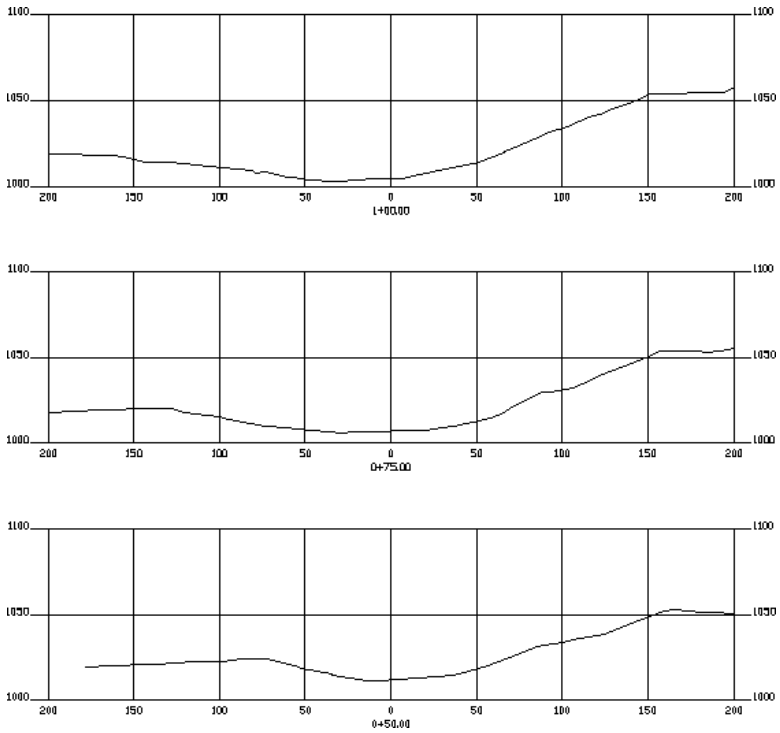
Select the OK button at the bottom of the dialog to begin plotting. For the Vertical Stack and Pick Location options, you are prompted to specify a starting point for the sections. If Sheet option was selected, another dialog appears to specify all the settings for sheet plotting, see details below.

Prompts

Select Starting Point for Row of Sections *pick a point*





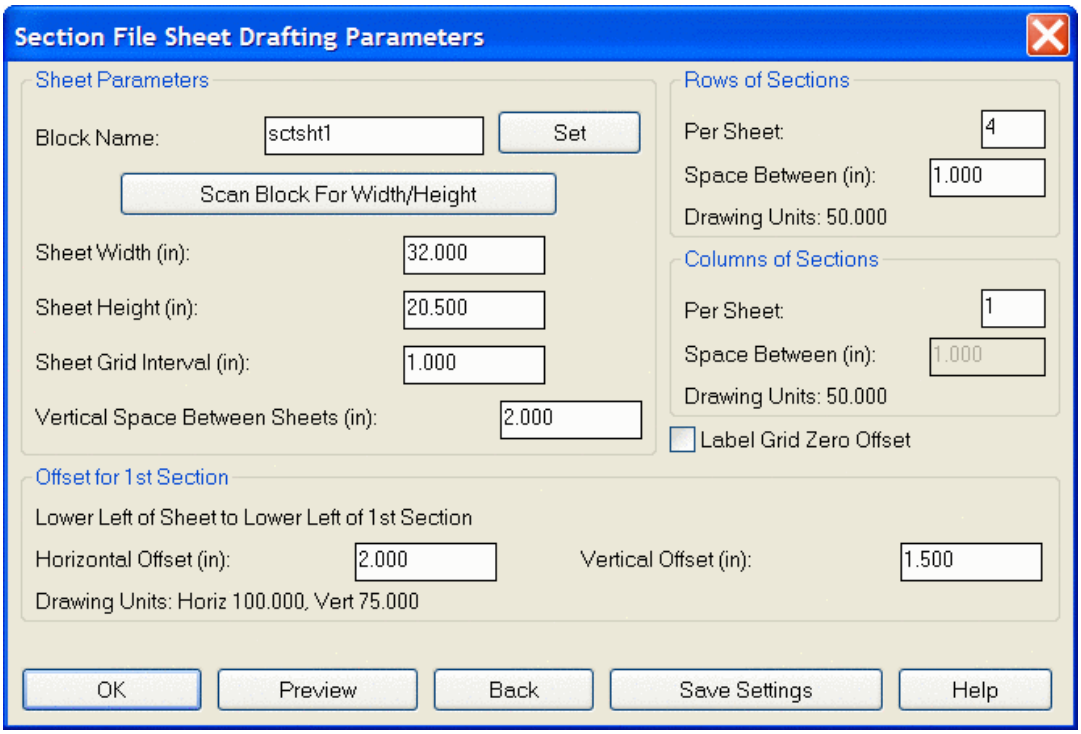


Vertical Stack layout

The Pick Location type of plotting has the following prompts:

Station> 4000.000 Min Elev> 462.849 Max Elev> 472.091
Change datum elev/<Select point that represents 0 offset elev 460.0>: *C*
Starting-Datum Elevation: *450*
The program scans the station data and determines the minimum and maximum elevations, and proposes a datum elevation. If you have pre-plotted a grid sheet and want to reference another local grid coordinate, then change the datum elevation appropriately.
Change datum elev/<Select point that represents 0 offset elev 450.0>: *pick a point*
Station> 4025.000 Min Elev> 463.332 Max Elev> 472.385
Change datum elev/<Select point that represents 0 offset elev 460.0>: *pick a point*

The program continues to prompt until the last station in the range specified is drawn. You can use the Cancel function ([Ctrl] + [C]) to stop plotting, if necessary. If you chose the Vertical Stack option you will be prompted for the starting point for the row of sections. If you selected the Sheets option, and press the OK button, the Section File Sheet Drafting Parameters dialog appears, allowing you to set up how you want the section sheets plotted.



Sheet Parameters
Block Name: Specify the AutoCAD drawing name that will be inserted for each sheet. The default is SCTSHT1 which is included with Carlson 2007, and is stored in the \SUP directory. You can use this or use a sheet block of your own design. The block should be drawn at a 1 = 1 scale since the program inserts it at the horizontal scale setting from the previous dialog.
Distance Between: Controls the distance from the bottom of one sheet and the bottom of the next.

Rows of Sections
Per Sheet: Controls how many sections will be stacked on top of each other on a sheet.
Distance Between: Controls how much space will be placed between the top of the last section plotted and the bottom of the next section. The distance between and other values in this dialog are in AutoCAD units. In our above example we are set to 20 horizontal scale so 20 would equal 1 inch when plotted. It is recommended that you set the

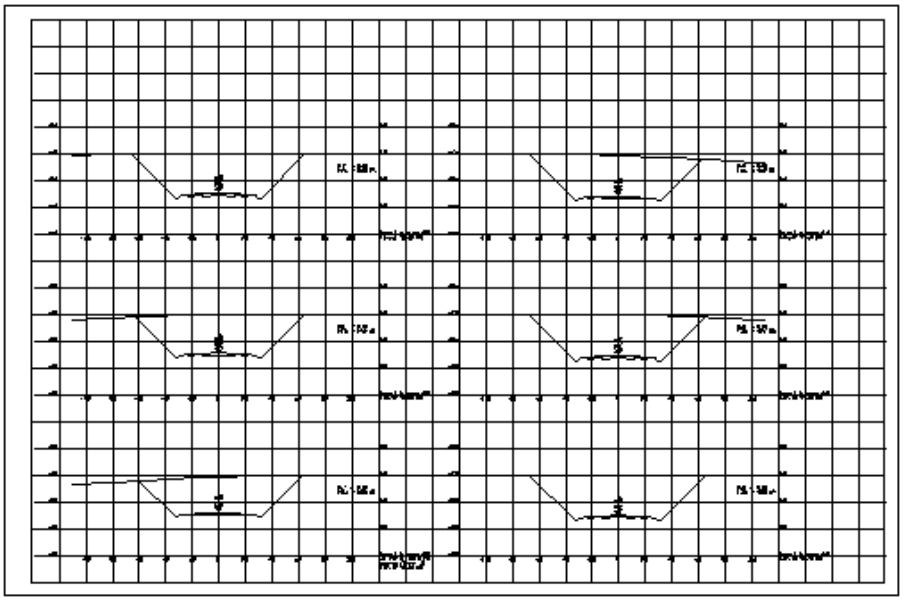
horizontal and vertical scales in the previous dialog before accessing the sheet parameters dialog so that reasonable defaults will be set automatically.

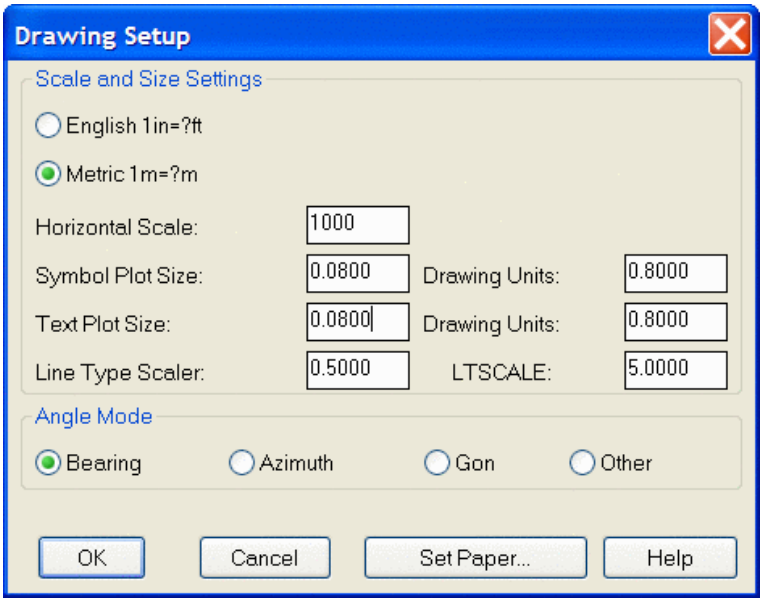
Columns of Sections
Per Sheet: Controls how many rows of sections will be plotted on each sheet.
Distance Between: Controls the distance that the rows of section will have between the centerline of the one section row and the next centerline of rows. This edit box can only be accessed if you have a number of columns greater than one. For example, if you want 15 inches between the columns, specify 300 (15 x 20).

1st Section Offset from
Lower Left of Sheet to CL: X and Y edit boxes allows you to specify where the first section of the first row will be placed relative to the lower left of the section sheet. In our example we specified 160 (8 inches at 20 scale) and 15 (1.5 inches at 20 scale). The Block SCTSHT1 has a half inch border before the 1st grid line and we want to plot starting at the second grid line, which is another inch from the bottom of the sheet. We want the centerline of the first section to be slightly left of the center of the sheet which is 33.5 inches wide so we specify 15 inches (300 at 20 scale).
Previous: This button allows you to return focus to the main dialog and make changes to settings or cancel the program. One thing to remember when plotting sheets with grid lines on them is to switch on the Text Only toggle on so you don't get duplicate grid lines.
Label Grid Zero Offset:
Save Settings: This button allows you to save all the parameters settings to a file so you can easily recall them for another project.
Load Settings: This button allows you to recall the settings saved with the option explained above.

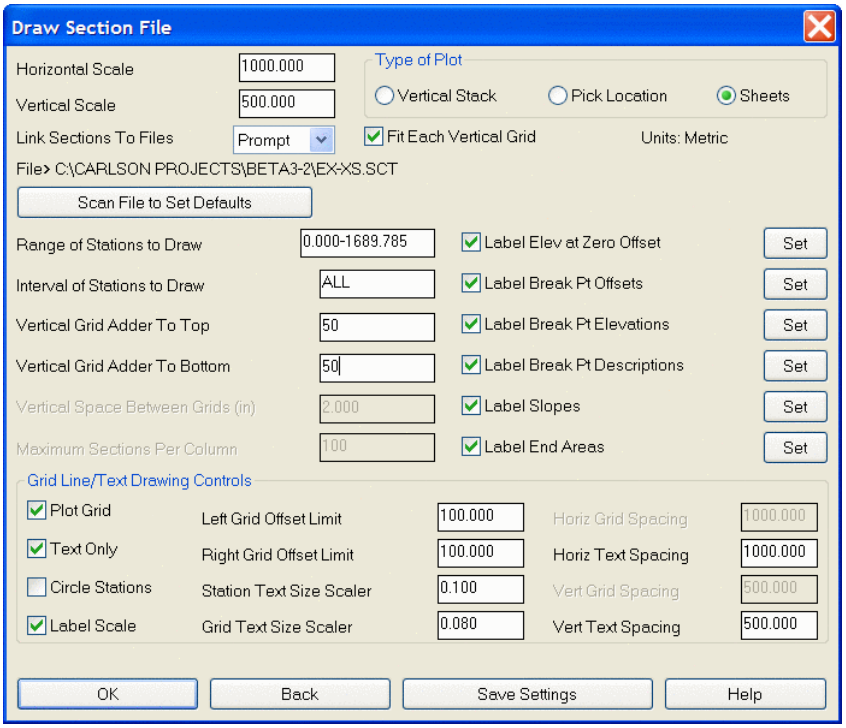
When you select the Layers button this dialog appears allowing you to specify the layer that the files are plotted on. If you are specifying a new layer to create, type the name into the edit box. If you want to select a layer that already exists from the layer list, then click on the Select... button to the right of the edit box. When you select the OK button the program prompts for the starting point for the row of sheets. The default is coordinate 0, 0 though you can select any point you like. With the settings shown in the example dialogs the sections would be plotted as shown below.

Drawing Metric Section Sheets
First, be sure that you are set to metric mode in *Drawing Setup* under the Settings menu. Then set the scales and spacing as shown in the dialog below. This example is 1:1000 scale. When the first dialog is set, click OK to reach the second dialog. There is a different block name for metric sections called schsht2.dwg. This file is located in the Carlson 2007 SUP directory. Choose the parameters for the second dialog as shown. In this case the sheets will have two rows and two columns of sections.

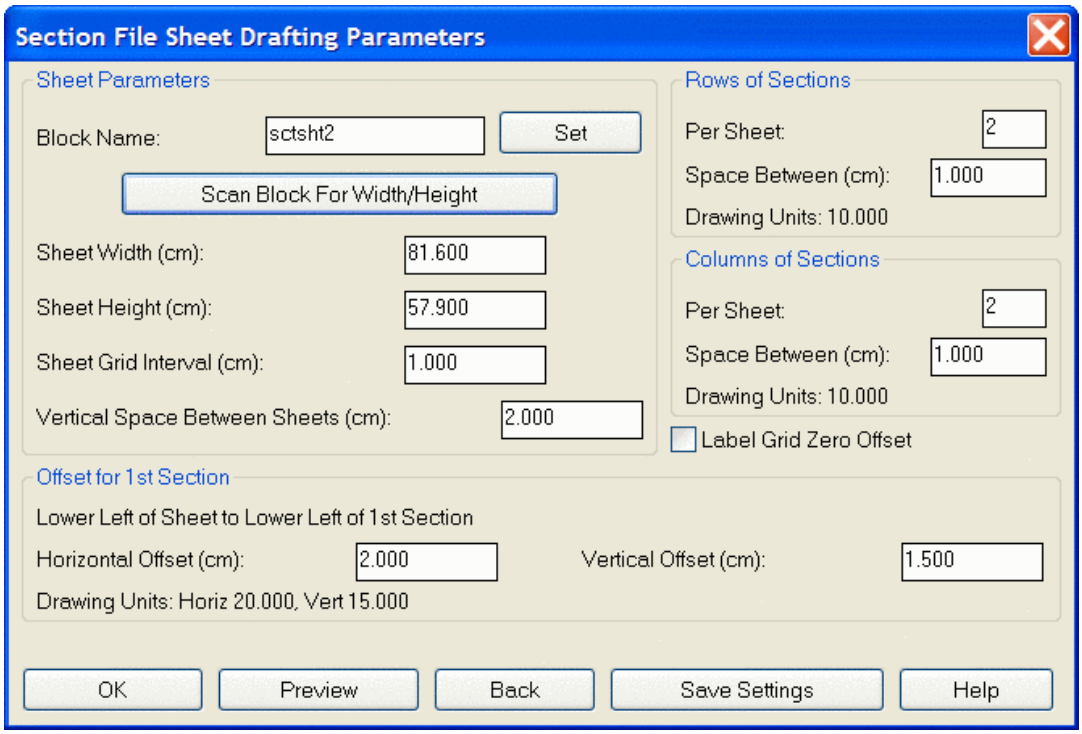




Drawing Setup dialog with metric 1m=?m setting from Inq-Set menu



First dialog with metric settings



Second dialog with metric settings

Keyboard Command: drawsct
Prerequisite: An .SCT file

Section Report

Function

This command generates a report of a section file for the specified stations. The information contained in the report is determined by the settings in the Section Report Options dialog box.

Decimal Places: Specify the display precision for stations and elevations.

Use Row-Column Report Layout: When checked, offsets are reported in columns. Example reports showing the difference are shown below.

```
==========================================
Row-Column Layout ON
==========================================
Section Report      05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

STATION

16+78.12   -260.00  -259.65  -244.64  -234.32  -213.99
           1401.30  1401.30  1400.92  1400.77  1400.62
           EXTRAPOL 0        0        0        0

==========================================
Row-Column Layout OFF
==========================================
Section Report       05/15/2002 18:31
File: C:/scadxml/DATA/125.sct

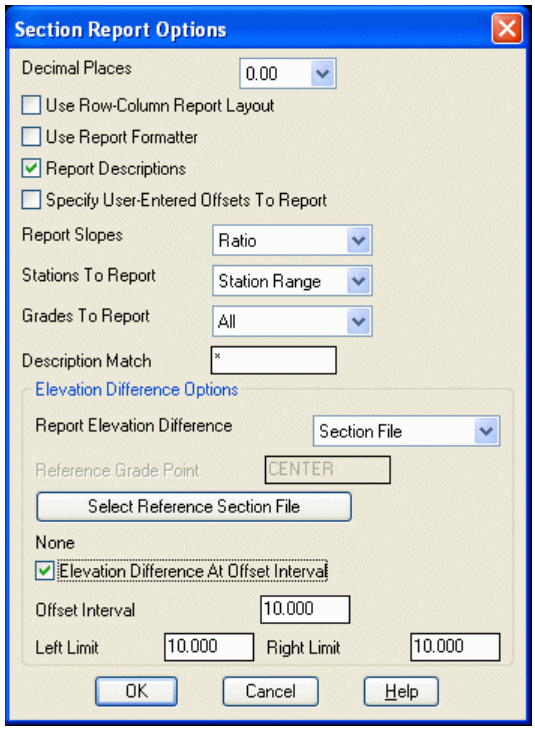
STATION: 16+78.12
-260.00    1401.30    EXTRAPOLATED
-259.65    1401.30    0
-244.64    1400.92    0
-234.32    1400.77    0
-213.99    1400.62    0
-209.00    1400.71    0
-185.27    1401.44    0
-183.67    1401.49    0
-183.33    1401.50    0
-158.34    1401.37    0
```

Use Report Formatter: Report output is directed to the Report Formatter which allows for custom reports, as well as being able to export the report to Microsoft Excel or Access

Report Descriptions: Controls whether the descriptions for each section point are reported.

Specify User-Entered Offsets To Report: This checkbox will use offsets you set in the report.

<>



Report Slopes: Will report the slope between section points.</> Specify how to report the slopes, either precent, ratio, or automatic. Automatic means that slopes less than 10% are reported in percent while greater slopes are reported as ratio.

Stations to Report: Specify either a range and interval of stations to report or enter each station one at a time.

Grades to Report: This applies to section files that contain subgrades. For these section files, you can choose which grades to report (top surface or subgrades).

Description Match: Can be used to report section points with a certain description. For example if you want a report of only the EP points, then enter a Description Match of EP.

Report Elevation Difference: Will report the section elevations as the elevation difference from a reference grade, or section file.

Reference Grade Point: Specify the reference grade ID. Only available if reference grade toggle is selected.

Section File: If section file is select to report the elevation difference from than you can select the section file to use and an offset interval.

Prompts

Section Report Options dialog

Starting station for report <0.000>: *Press Enter*

Ending station for report <1147.478>: *Press Enter*

Station interval (A for All) <100.0>: *Press Enter*

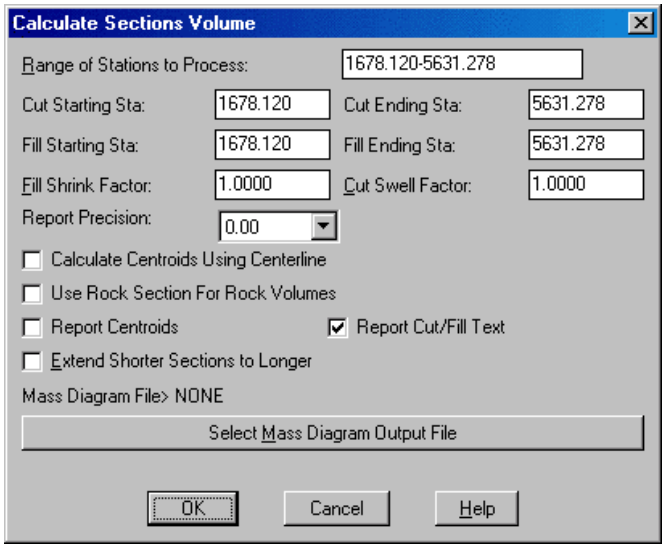
Prerequisite: A Section File (.sct)

Keyboard Command: sctrprt

Calculate Sections Volume

Function

This command will read two section files and compute the cut and fill end areas and volumes. This command computes the sections volume in the order they appear in the file. If you need to sort the stations in sequential order use the *Input-Edit Section File* command. Begin by selecting the base section file then the final section file. After specifying the input files the Calculate Sections Volume dialog appears.



Range of Stations to Process: Specify the range of stations to process. Separate stations with a hyphen as shown.

Cut/Fill Starting/Ending Sta.: Volumes are calculated using end areas between the range of stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station in the range. Likewise the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station.

Fill Shrink/Cut Swell Factor: Allows you to specify a value that the volume calculated will be multiplied by.
Report Precision: Specify the display precision for the report.

Calculate Centroids Using Centerline: When checked, the program will calculate the centroids using a centerline (.CL) file. you will be prompted to select the centerline file.

Use Rock Section for Rock Volumes: When checked, you will be prompted to select a third section (.SCT) file that will be used to calculate rock quantities.

Report Centriods: Specify whether or not to report centroids.

Report Cut/Fill Text: Specify whether or not to report cut/fill text.

Extend Shorter Sections to Longer: If checked, shorter sections are lengthened to the same left and right offset extents as the corresponding longer sections.

Select Mass Diagram Output File: Specify an optional profile (.PRO) file for haul data output.

Prerequisite: Two section (.SCT) files.

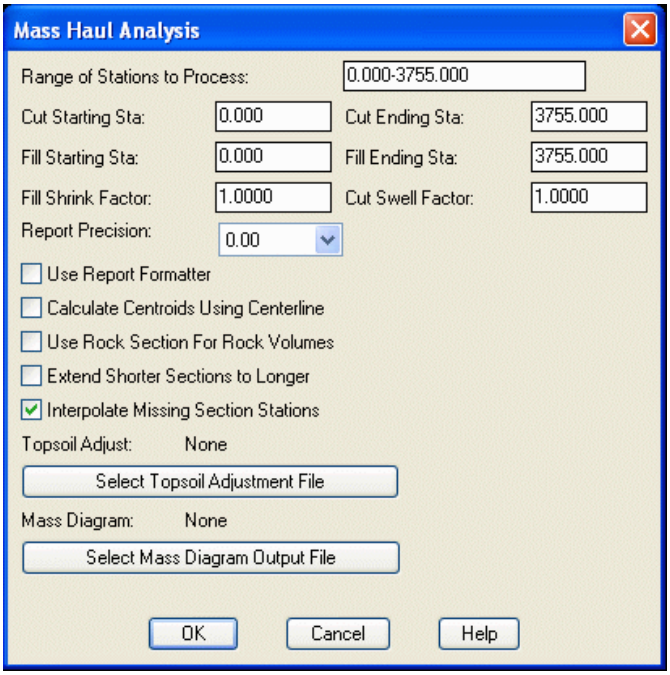
Keyboard Command: calcsct

Mass Haul Analysis

Function

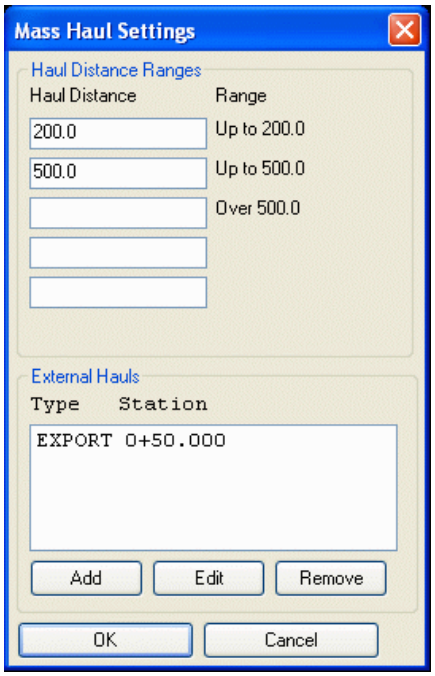
This command will determine the volume and haul distance for each group of net cut and net fill station ranges along a road. The program calculates the optimized cut to fill movements so that the total volume-distance moved is minimized.

You will first be prompted to select the Existing Ground section file and the Design Surface section file. These two surface files will be used to determine the Mass Haul quantities. If you do not have either of these files, you can create them using the different Create Sections commands under Roads. After you selected your section (.sct) files, the following dialog will appear.



Range of Stations: The program will pick up the range of stations determined by your section files. In this field, you can modify the range of stations to process.
Cut/Fill Starting/Ending Stations: The Cut and Fill Starting and Ending Stations are for tapering the end areas at the start and end of the section range down to zero beyond the station range.
Shrink/Swell Factors: The Shrink Factor is multiplied by the fill quantities and the Swell Factor is multiplied by the cut quantities.
Report Precision: This setting controls the number of decimal places to use in the report.
Use Report Formatter: The Report Formatter will allow you to customize the information reported by the Mass Haul Analysis.
Calculate Centroids Using Centerline: This option will find the center offset for each Cut/Fill area and use a centerline to adjust the station interval along curve segments for the end area volumes.
Use Rock Section For Rock Volumes: This option will use a third section file for reporting rock cut quantities.

Extend Shorter Sections to Longer: This option will find your longest section and match the length of all your other sections to it.
Interpolate Missing Section Stations: Toggle this on to interpolate any missing stations so that the Mass Haul report can use all the stations.
Topsoil Adjustment: This will apply a Topsoil Removal/Replacement definition from the Template Adjustments to adjust the sections.
Mass Diagram: This will create a Mass Diagram of the cut/fill balance by station. This data is stored in a profile file (.pro) format file, and you can use Draw Profile to draw it.



Mass Haul Settings

The Haul Distance ranges are for reporting the cut to fill volume movements by the different haul distance ranges. The purpose is to evaluate how far the cut has to be moved, and the haul distance ranges can be used to separate the distances for different types of equipment. The External Hauls can be used to specify the stations along the road for borrow pits or dump piles. The program will use volume from these extrenal hauls when the cut/fill of the road does not balance.

Mass Haul Analysis

Existing Section> C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo2-og.sct
Final Section> C:\Documents and Settings\Todd Carlson\Desktop\Takeoff\Drawings\demo2-fn.sct
Haul Distance Ranges
Net Cut Net Fill Total In Sta Haul 0 200 Over
Station Station Cut(CY) Import Fill(CY) Export Volume Volume Volume 200 500 500 Avg Haul
0+00.000 1+00.000
1+00.000 2+50.000 1251.729 0.000 1251.729 0.000 1251.729 1127.595 124.134 124.134 0.000 0.000 153.135

12+00.000 10+50.000
13+80.000 12+00.000 887.367 0.000 887.367 0.000 887.367 239.938 647.429 542.651 104.777 0.000 179.887

13+70.000 15+05.340
13+90.000 15+23.200 95.633 0.000 95.633 0.000 95.633 51.559 44.074 44.074 0.000 0.000 137.235

13+80.000 14+20.000
14+20.000 14+50.000 216.434 0.000 216.434 0.000 216.434 96.872 119.563 119.563 0.000 0.000 38.560

14+60.000 14+50.000
14+80.000 14+60.000 43.333 0.000 43.333 0.000 43.333 18.620 24.712 24.712 0.000 0.000 11.818

14+70.000 15+00.000
15+00.000 15+14.270 82.194 0.000 82.194 0.000 82.194 29.738 52.456 52.456 0.000 0.000 24.395

Total: 9808.744 0.000 69788.70 0.000 74189.76 66786.40 7403.360 1288.814 930.060 783.423 130.579

Prerequisite: A Section Alignment File and Existing and Road Sections

Keyboard Command: masshaul

Calculate End Area

Function

This command allows the user to select two polylines representing an existing grade section and a final grade section and calculate the end area. The area calculated can be drawn at a user specified point. Optionally the command writes the stations cut and fill to an earthwork (.EW) file that can be printed/displayed by the *Print EW File Report* command.

This command starts with the Calculate End Areas dialog

Horizontal Scale: Specify the horizontal scale of the existing cross section.

Vertical Scale: Specify the vertical scale of the existing cross section.

Text Scale: Specify the text size scaler, this value is multiplied by the horizontal scale to determine the final text height.

Station Interval: Only available if Write Results to EW File is toggled on. Allows you to specify the station interval that the station prompting will default to as you select the polyline/sections for computation.

Extend Shorter Ends to Longer:

Write Results to EW File: When checked, the results will be written to an earthwork (.EW) file. You may select a New file or choose to append/revise an existing file.

Prompts

Select existing polyline: *(select polylne)*

Select final polyline: *(select polylne)*

Total Fill Area= 166.77

Total Cut Area = 247.95
Station number <0.000>: *[Enter]*

Pressing [Enter] selects the default station 0+00. If the station does not exist in the file it will be added. If it does it will be revised.

Select point for End Areas label: *(pick Point)*

Select existing polyline: *[Enter]*

Pressing [Enter] ends the prompting and the command sorts the file. Because of the sort the user can select stations in any order and they will be arranged in ascending order for proper volume computation.

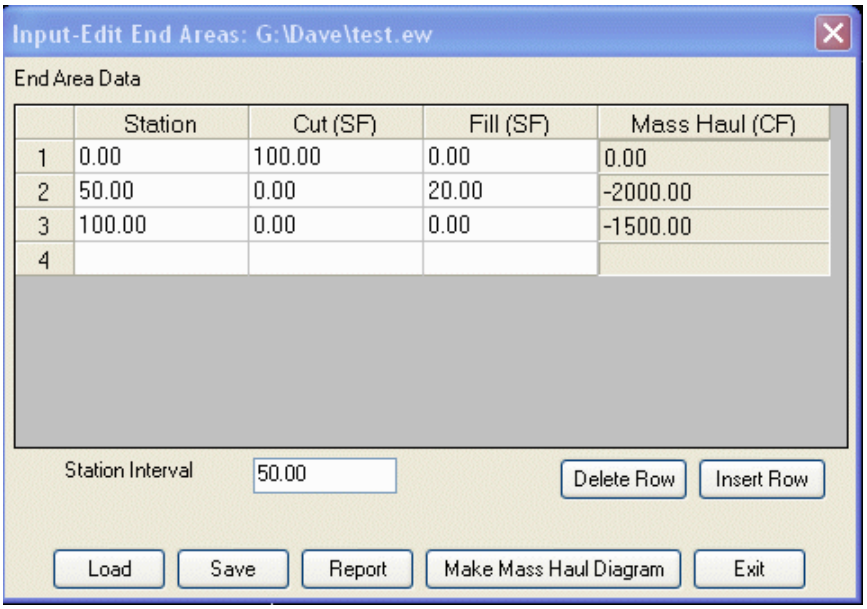
Prerequisite: Plot the existing grade and final grade polyline/section.

Keyboard Command: endarea

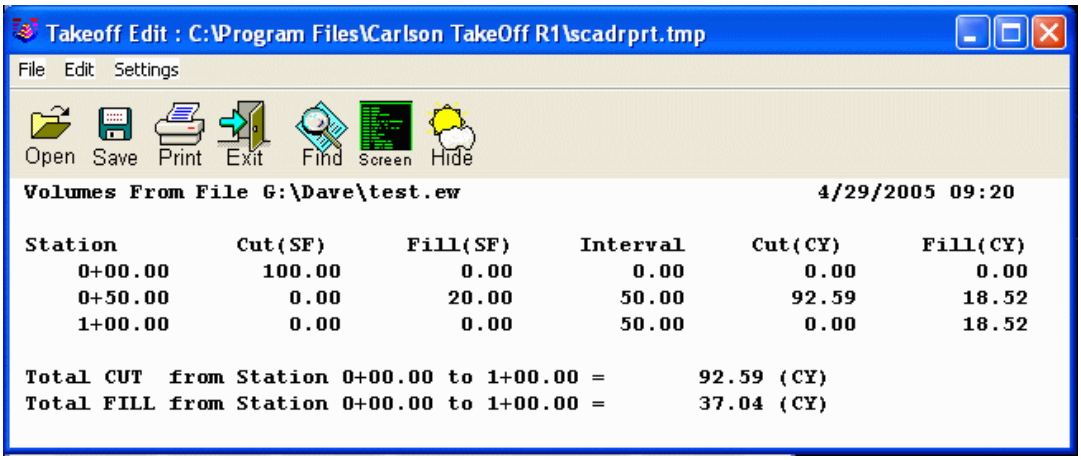
Input Edit End Area File

Function

This command allows you to enter in Stations, Cut (SF), Fill (SF) data to calculate a project's Mass Haul (CF).

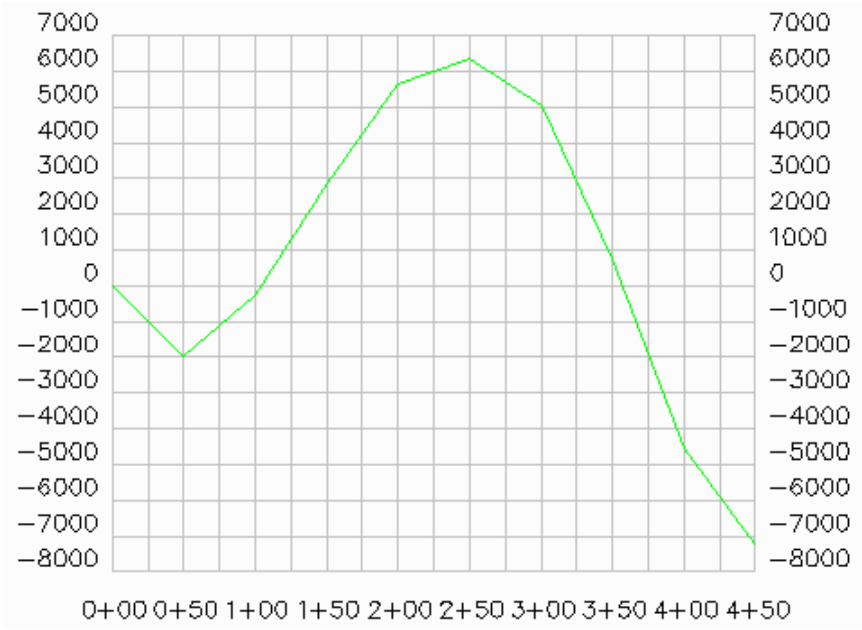


You can set the Station Interval and Delete or Insert a Row at your discretion. The Report button will show you the Cut/Fill at each station as well as the Total Cut/Fill.


Takeoff Edit : C:\Program Files\Carlson TakeOff R1\scadrprt.tmp
File Edit Settings
Open Save Print Exit Find Screen Hide
Volumes From File G:\Dave\test.ew 4/29/2005 09:20
Station Cut(SF) Fill(SF) Interval Cut(CY) Fill(CY)
0+00.00 100.00 0.00 0.00 0.00 0.00
0+50.00 0.00 20.00 50.00 92.59 18.52
1+00.00 0.00 0.00 50.00 0.00 18.52
Total CUT from Station 0+00.00 to 1+00.00 = 92.59 (CY)
Total FILL from Station 0+00.00 to 1+00.00 = 37.04 (CY)

Make Mass Haul Diagram

Use the Make Mass Haul Diagram to create a .pro file of your data. Then run the command Report Profile to create a report of of your mass haul data or run Draw Profile to create a diagram like the one below.



Prerequisite: End Area Data

Keyboard Command: ewedit

Print Earthwork File Report

Function

This command is used to display/print an earthworks (.EW) file. This file can be generated several TakeOff com-

mands such as *Calculate End Area* or *Digitize End Areas.* A standard file selection dialog prompts you for the .EW file, then the report is displayed in the Standard Report Viewer.

Volumes From File c:\scadxml\data\main.ew 05/16/2002 07:55

Station Fill(sf) Cut(sf) Interval Fill(cy) Cut(cy)

34+00.00 0.0000 24.7082 0.0000 0.0000 0.0000

35+00.00 0.0000 75.0246 100.0000 0.0000 184.6904

36+00.00 0.0000 29.1810 100.0000 0.0000 192.9733

37+00.00 0.3867 1.4916 100.0000 0.7161 56.8011

38+00.00 31.1798 0.0000 100.0000 58.4565 2.7622

39+00.00 57.6465 0.0000 100.0000 164.4931 0.0000

Total FILL from Station 34+00.00 to 39+00.00 = 223.6657 (cy)

Total CUT from Station 34+00.00 to 39+00.00 = 437.2270 (cy)

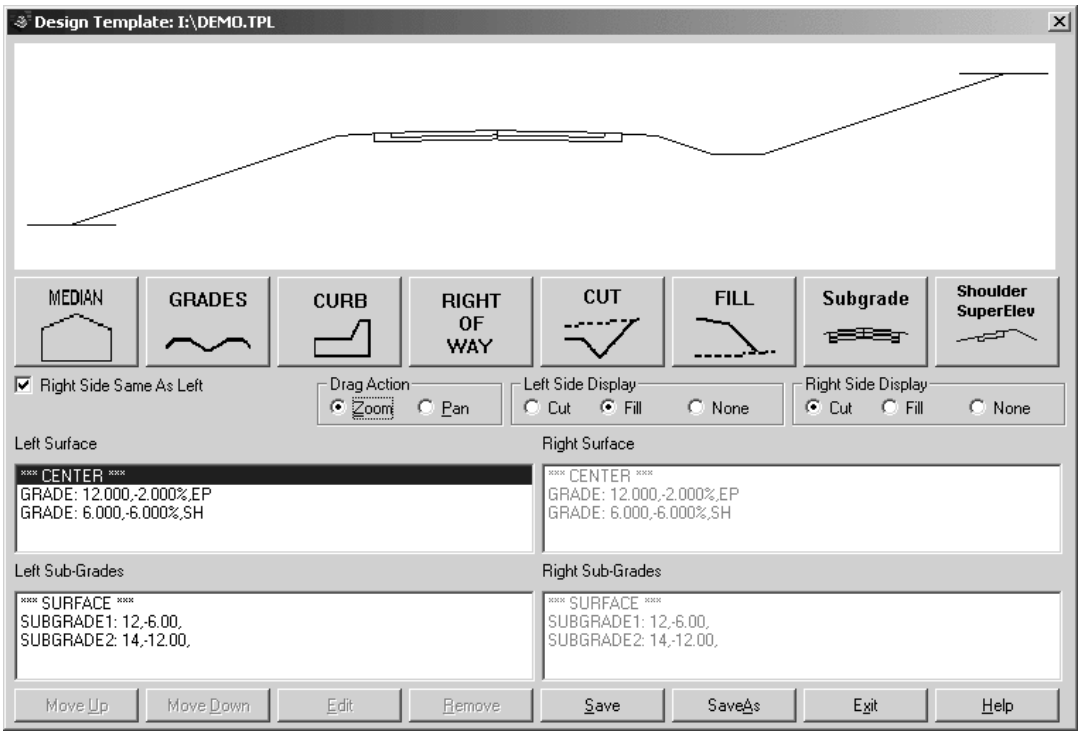
Prerequisite: Prepare .EW file.

Keyboard Command: ewreport

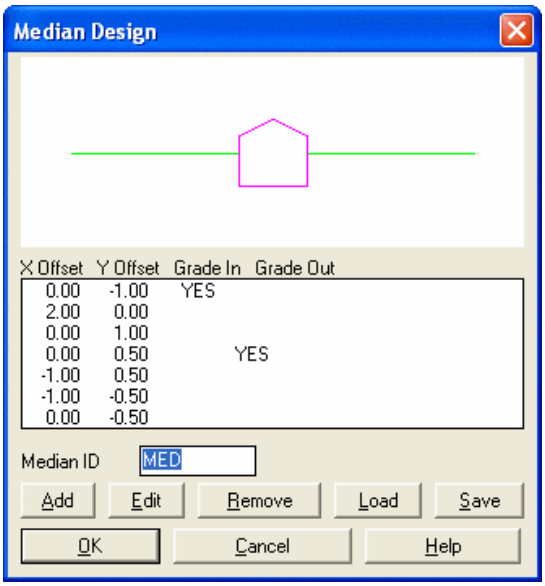
Design Template

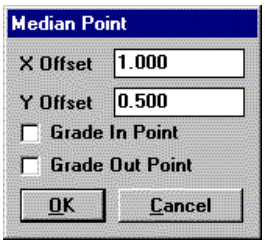
Function

This command creates a template definition file (.TPL file). The template file can then be applied in the *Process Road Design, Draw Typical Template* or *Design Pad Template* commands. The template is designed using the dialog shown below. The top portion shows the template as you create it. In the middle is a row of icons which are the building blocks of the template. They can be chosen in any order by picking on the icon. In the bottom of the dialog are four list boxes that list the elements of the template. The surface elements are listed in order starting from the center. The subgrades are listed from top to bottom order. To add a template element, highlight the position in the list above where to insert the element. Then pick one of the element icons. To change the order of an element, highlight the element and pick the Move Up or Move Down buttons. The Edit button edits the dimensions of the highlighted element. The Remove button erases the highlighted element from the list. There is no limit to the number of surface or subgrade elements. Note that there is a Right Side Same as Left option. When active this option only requires template design for the left side and will automatically mirror the design for the right side.

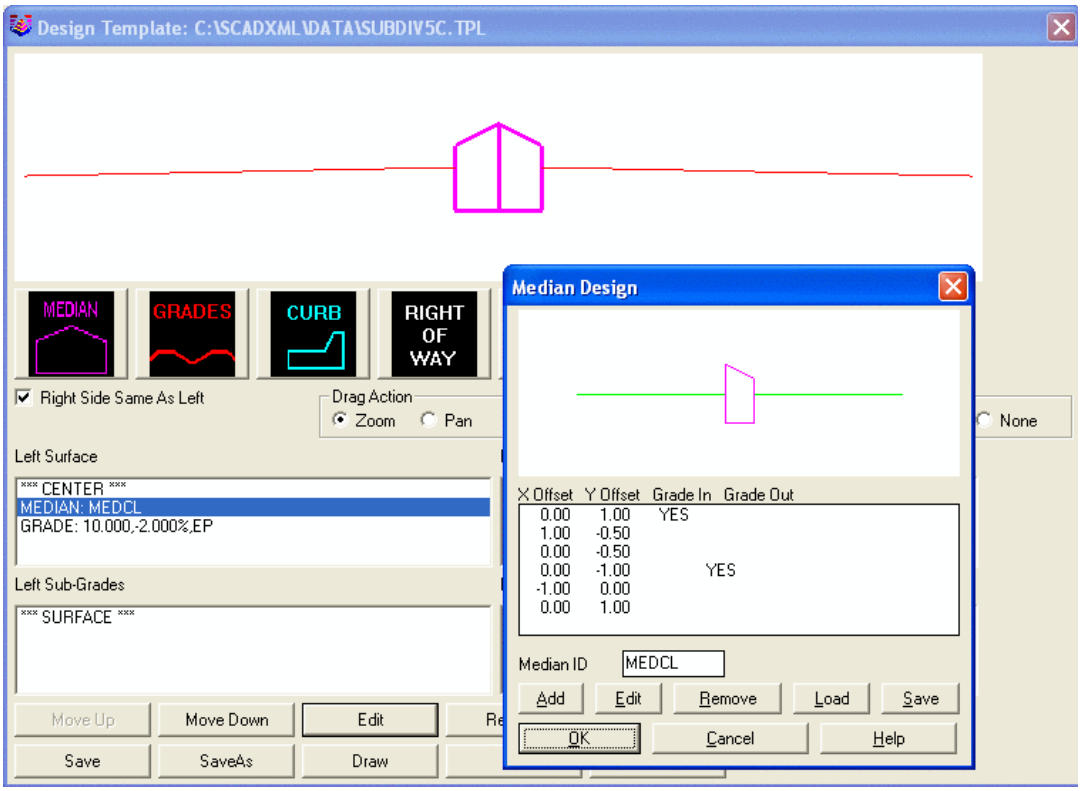


The template surface can be composed of three types of elements: medians, grades and curbs. The median is a flexible closed figure defined in a clockwise direction. Each median point consists of an X and Y offset. The median must be closed and the program will automatically create the closing segment. In the Median Design dialog, the median is shown in the top display and bottom has a list of median points. The display shows the median in magenta and the grade lines in and out in green. For the display the grade in comes from the left and the grade out goes to the right. The median must define the Grade In point which is the point that ties into the incoming surface grade. Also the Grade Out point must be specified for where the surface grade continues out from the medianThese Grade In and Grade Out points emanate from the starting or "from" position in the coordinate dialog where they are specifiedSince a single median must be placed on the left or right side (and is typically not used symmetrically with right side same as left), you will need to offset the template centerline one-half the median width within the command Process Road Design in order to center the medianYou will also have to move the "C/L" designation, to obtain centering, when using Draw Typical Template.

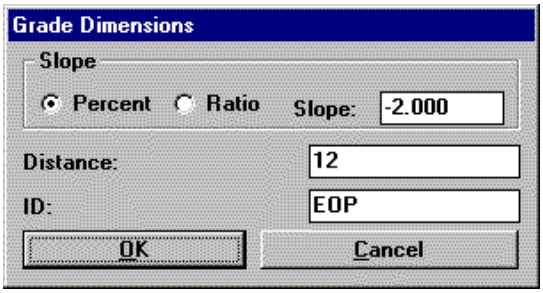




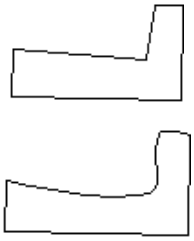
You can design a median for "mirroring" to create a centered effect, as shown below. The only negative to this method is the appearance of a vertical line in the median plot. Medians can be saved and loaded for re-use in other templates.



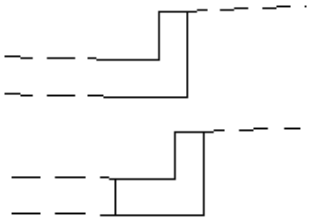
Surface grades can be entered by selecting the Grades icon which brings up the dialog shown. Downhill slopes are negative and the Distance is the horizontal distance. The text ID serves 4 purposes: (1) The ID will be applied as a description to all final template points generated in the form of a coordinate (.CRD) file, (2) The ID can be used as a design point, as in EP+5 indicating 5 feet or meters right of edge of pavement, (3) Points of common ID may be connected by 3D polylines as an output option of *Process Road Design* and (4) Quantities can be generated with reference to the ID and material (gravel, concrete, etc.) entered elsewhere within this command.



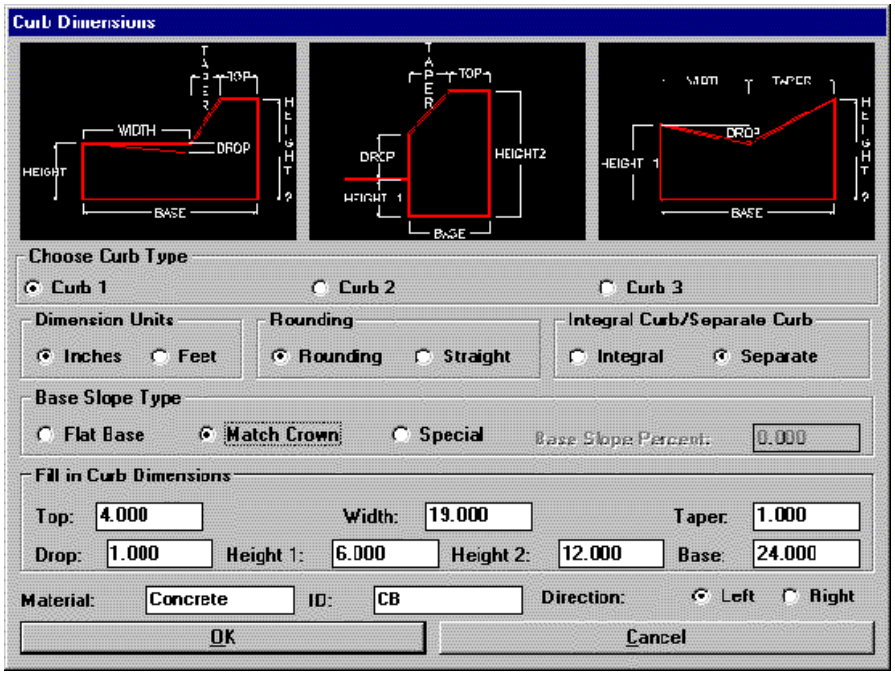
To add a curb, select the Curb icon. The dialog box below appears where you can fill in the curb dimensions. There are three curb types to choose from. The curb dimensions can be specified in feet, inches or meters in metric mode. The Rounding option will smooth the surface of the curb which only shows when the template is applied in commands such as *Process Road Design*. The Integral/Separate option determines whether to draw the front line of the curb to separate the curb from the subgrade. For example, fully concrete pavements that contain a curb would be drawn with the "integral" curb optionThe slope of the curb can either be flat, set to the slope of the incoming grade or set to a user-specified slope. The material name is used in the *Process Road Design* report

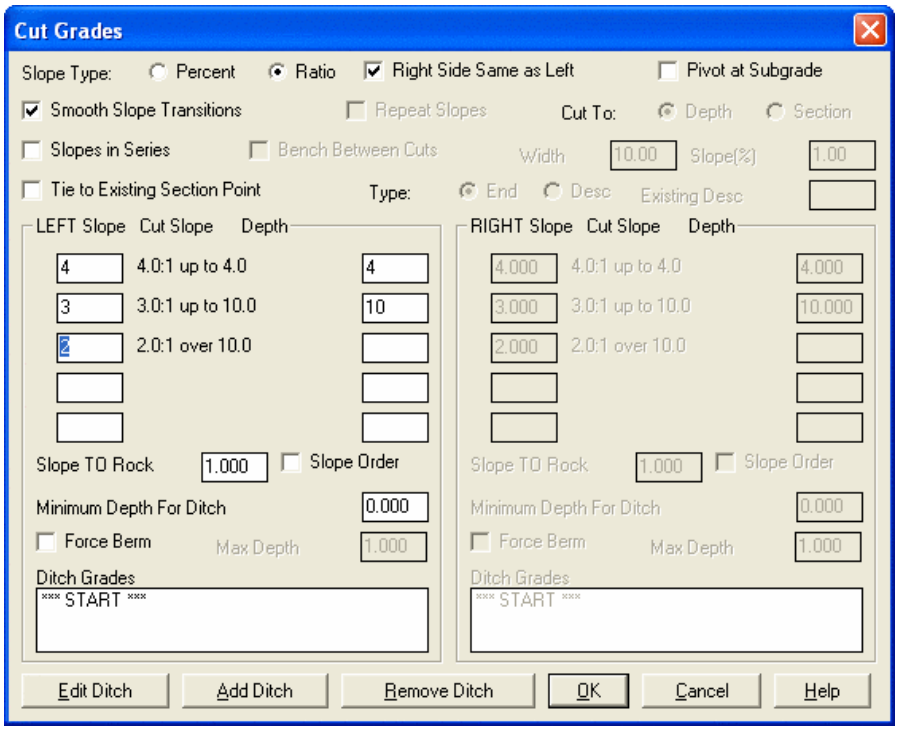


Straight & rounded curbs



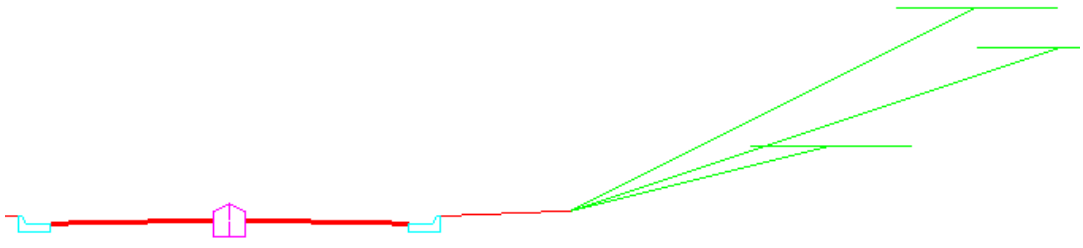
Integral and separate curbs




Cut Grades
Slope Type: Percent Ratio Right Side Same as Left Pivot at Subgrade
Smooth Slope Transitions Repeat Slopes Cut To: Depth Section
Slopes in Series Bench Between Cuts Width 10.00 Slope(%) 1.00
Tie to Existing Section Point Type: End Desc Existing Desc
LEFT Slope Cut Slope Depth
4 4.0:1 up to 4.0 4
3 3.0:1 up to 10.0 10
2 2.0:1 over 10.0
Slope TO Rock 1.000 Slope Order
Minimum Depth For Ditch 0.000
Force Berm Max Depth 1.000
Ditch Grades
*** START ***
RIGHT Slope Cut Slope Depth
4.000 4.0:1 up to 4.0 4.000
3.000 3.0:1 up to 10.0 10.000
2.000 2.0:1 over 10.0
Slope TO Rock 1.000 Slope Order
Minimum Depth For Ditch 0.000
Force Berm Max Depth 1.000
Ditch Grades
*** START ***
Edit Ditch
Add Ditch
Remove Ditch
OK
Cancel
Help

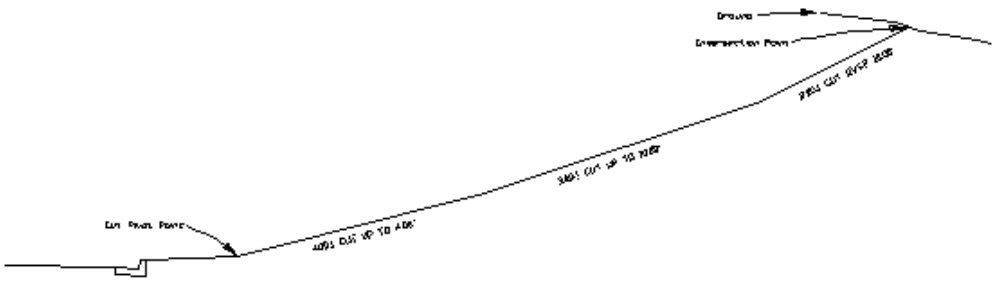
To specify cut treatment, pick the Cut icon. There is room to specify up to five cut slopes which can be slopes in series or slopes to use at different depths. In a simple case of one cut slope, you can just enter the one slope value and leave the depth and other slope boxes blank. For Slopes in Series, each slope is used up to the specified depth until an intersection with the ground. If the intersection is not reached by the first slope, then the next slope continues from where the first ended. If you have more than five slopes, pick the Repeat Slopes option which will repeat the sequence of entered slopes until the ground is reached. The Bench Between Cuts option allows you to enter a bench width and percent slope to be inserted between each cut slope. Besides running the cut slopes to specific depths, the Cut To Section option can be used to have each cut slope intersect a surface from a section (.sct) file. With Cut To Section on, the Process Road Design command will prompt for these cut slope section files. For example, this Cut To Section option could be used when you have a cut bench that occurs at a set elevation but different cut depths as the road profile changes. In this case, you could create a section (.sct) file at this set bench elevation.

With Slopes in Series off, just one of the slopes is used depending on the depth. For example, set the dialog as shown to use 4 to 1 slopes at depths up to 4 feet, 3:1 up to 10 and 2:1 if deeper. The effect is 4:1 if shallow and, by contrast, 2:1 if the fill is deep. The Smooth Transitions option will gradually transition the slopes from one range to the next. In this example, if the depth is 5 feet the slope will be between 4:1 and 3:1. The graphic in the Design Template dialog will explicitly show slopes in series versus individual slope depending on setting (shown below are individual slopes, with slopes in series off):



The Pivot at Subgrade option will position the cut pivot point where the bottom subgrade intersects the template grade. The ditch or upslope conditions will then occur from this special subgrade "daylight" pivot point, instead

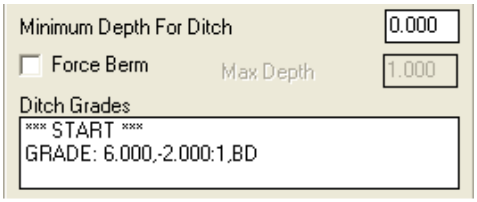
of from the outer shoulder surface pivot point. The Tie to Existing Point will draw the cut slope from the cut pivot point to either the outside offset-elevation or an offset-elevation point with a specified description from the existing section file. This method is used when survey crews take sections and designate the specific slope tie points.

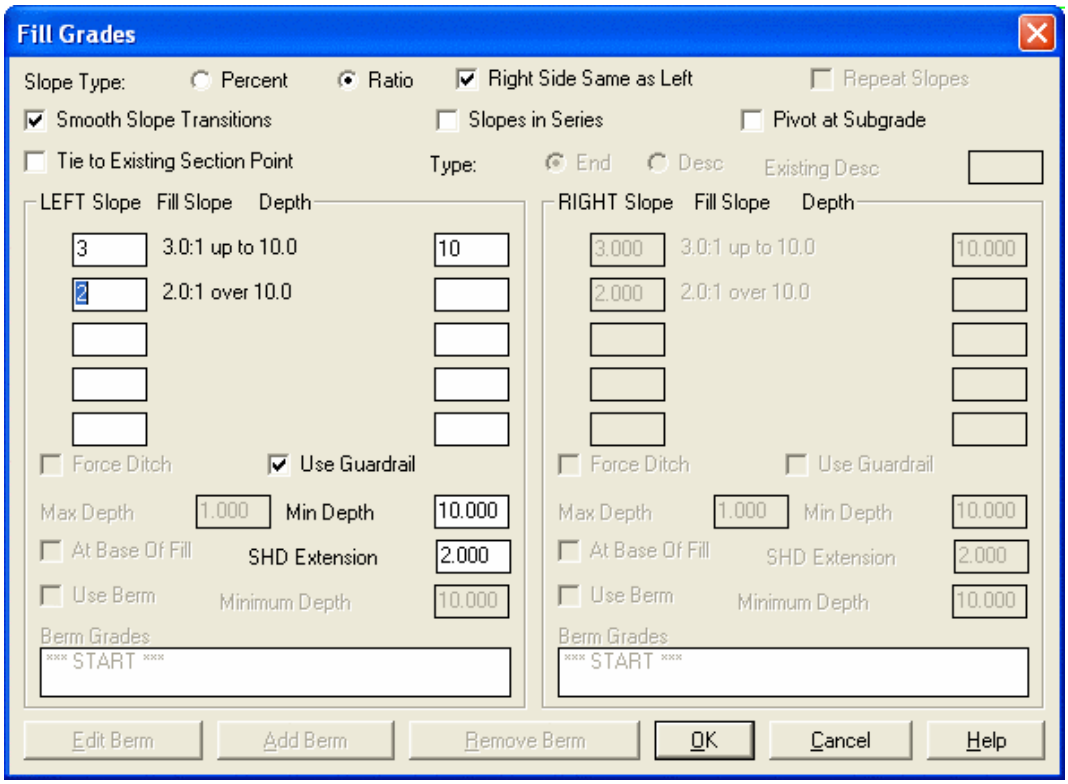


Three cut slopes in series

The Slope to Rock applies in Process Road Design when using a Rock Section File. There are two slope order modes for rock slopes: Slope TO Rock and Slope FROM Rock. For the Slope TO Rock mode, the cut slope will be the Slope To Rock up to the rock surface. After reaching the rock surface, the regular cut slopes apply. For the Slope FROM Rock mode, the regular cut slopes apply up to the rock surface. Then from the Slope From Rock applies from the rock surface to the ground surface.

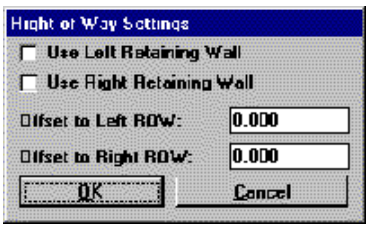
Ditch Grades can be inserted prior to the application of the cut upslope. For curb and gutter roads, there is typically no ditch. But for roads with drainage downhill to the outside and no curbs, ditches are typically used in cut conditions. The Ditch Grades list contains each ditch grade in order from the regular template. Any number of ditch grades can be added by picking the Add Ditch button. To create a V ditch, add just one ditch grade such as slope ratio -1, distance 1. This makes one side of the V. The pivot point for the cut slopes will be the bottom of the V and the other side of the V will be made by the cut upslopes. For a ditch with a flat bottom, you could have two ditch grades such as slope ratio -2, distance 4 and then slope percent 0, distance 2. If a minimum depth for ditch is entered, no ditch will be applied unless the cut exceeds that depth. The Force Berm will apply the Berm (defined using the Fill icon) in cut instead of a ditch up to a certain depth of cut.





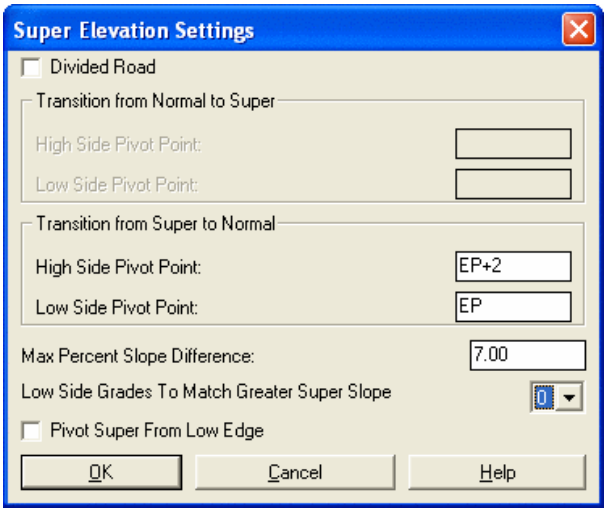
Fill treatment is similar to cut. Up to five slopes for different depths can be specified. Slopes in Series and Smooth Transitions work the same way as cut. Berm Grades are the fill equivalent to Ditch Grades. Fill treatment does have some extra options. Guardrail Expansion will extend the last template surface grade the specified Shoulder Distance when the fill is greater than the Min Depth. The Force Ditch option has two different methods to apply the Ditch Grades from the cut definition. With "At Base Of Fill" on, Force Ditch creates the ditch where the fill slope hits existing ground. With "At Base of Fill" off, the Force Ditch method applies the ditch grades from the template pivot point. The Minimum Depth for Berm Grades will only draw the Berm Grades when the fill depth is greater than the specified value.

The Right of Way icon brings up the dialog shown which allows you to specify whether to use a retaining wall to keep the cut/fill slopes from crossing the right of way. The right of way data is stored in a centerline file (.cl file) as stations and offsets for the left and right sides of a centerline. When the retaining wall option is active, the cut or fill slope will go at the design slope up to the right of way and then the slope will tie into the ground by going straight up or down. Without the retaining wall option, the cut or fill slope will become steeper in order to tie into the ground at the right of way. For example, if the cut slope is 50% but this slope ties into the ground past the right of way, then the slope will be modified to something steeper such as 65%. The Offset ROW options will force the tie in the offset distance before the right of way.



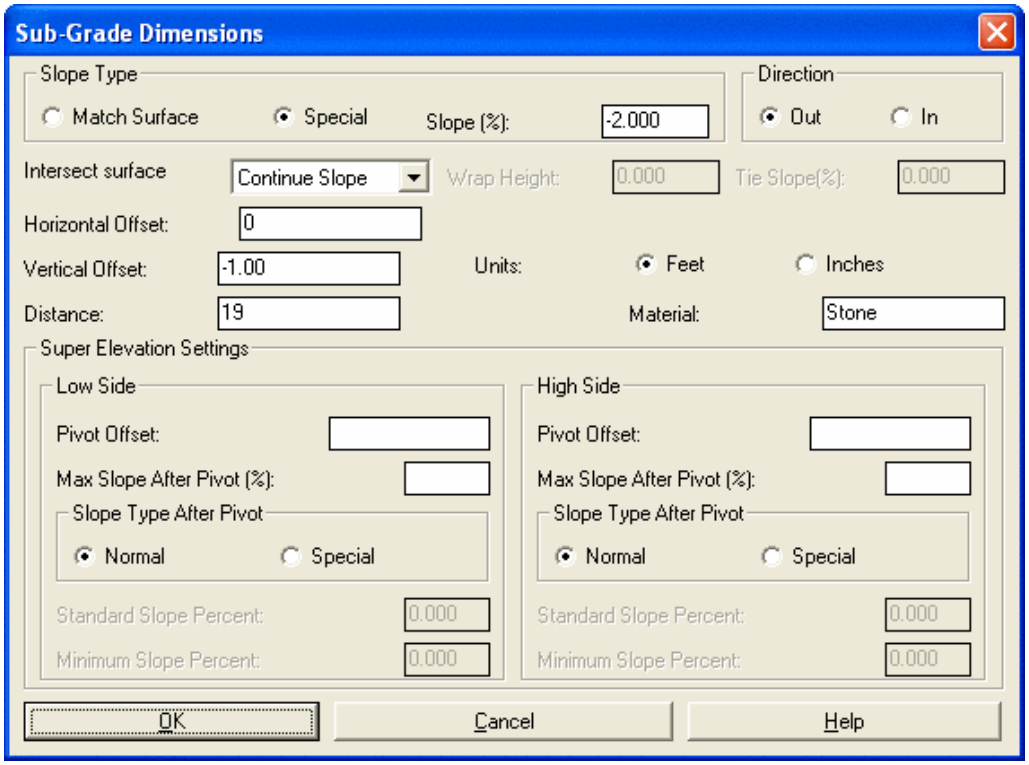
The Shoulder Super Elevation icon specifies where on the template the slopes will transition between super elevation slopes and normal slopes. The transition point is identified under Pivot Point by the template id for the grade, curb

or median. Note that the pivot point can be specified as an ID plus a distance as in "EP+2". Starting from the center, the template grades will be in super up through this template segment. For example, based on the template shown in the first dialog of this command, the EOP Pivot Point the Super Elevation Settings dialog will create the first EOP grade in super while the curb and grade S will be at normal grade. The High and Low Pivot Point options allow for different transition points depending on which side is raised by the super elevation. The Max Percent Slope Difference is the maximum difference between the super elevation grade and the normal grade at the pivot point. For example with a Max Percent Slope Difference of 7%, if the super elevation grade is 6%, then the slope after the pivot on the high side will be -1% even if the normal design slope is steeper than -1%. If the grades do not start from the center in super, then the Divided Roads option can be used. With this option, the grades start from the center as normal and then transition to super at the Normal to Super Pivot Point.



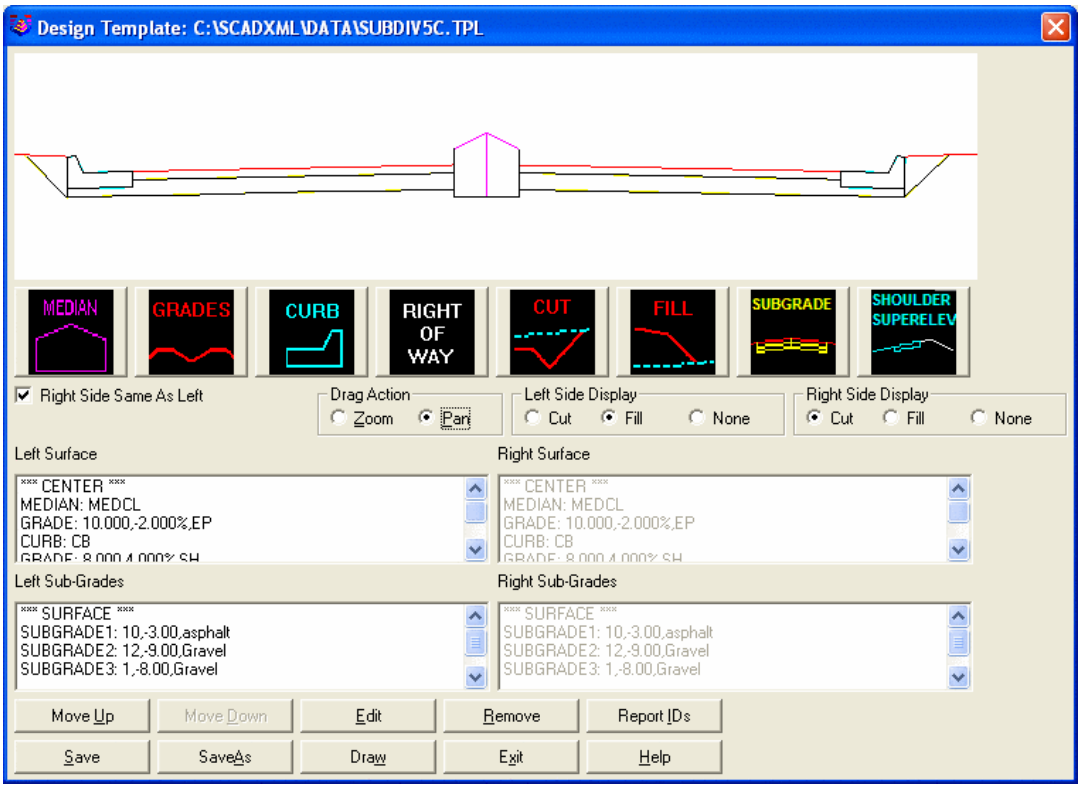


Example of super elevation of 4% to the right for a divided road with a Max Difference of 7%. The normal template is shown above. The Normal to Super Pivot Point is MED and the Super to Normal Pivot Point is EP. The result is that the EP segment is in super and the SH and MED segments are at normal slope. On the left, the SH segment is at the normal -10%, the EP segment is at the super elevation slope of -4% and the MED segment wants to be at 4% but ends up at 3% because this meets the Max Difference requirement. On the right side, the MED segment starts at the normal -4%, then the EP segment transitions into the super -4% and then the SH transitions back to normal which results in a 3% slope because of the Max Difference requirement.

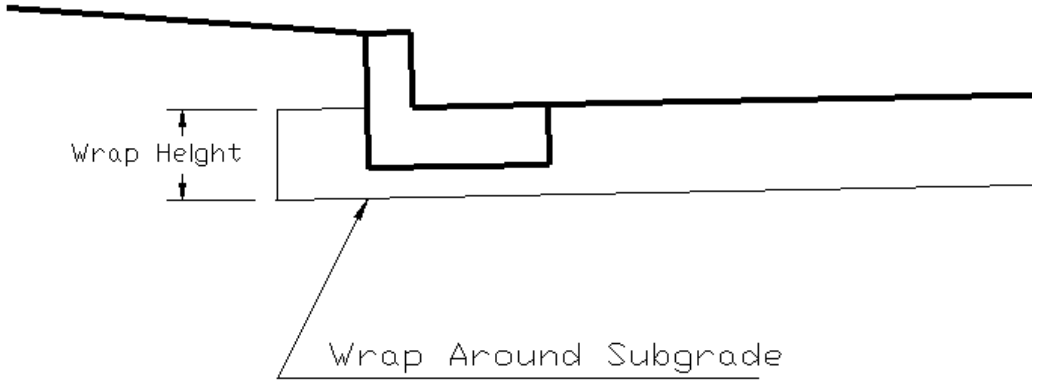


To add subgrades click the SubGrades icon which brings up the dialog shown. The subgrades are areas below the template surface. There can be any number of subgrades stacked one below another or side by side.

The subgrade starts from the surface at the distance from the center set under Horizontal Offset. To start from the centerline, enter 0 in Horizontal Offset. First the subgrade moves straight down from this Horizontal Offset. The depth down is specified in Vertical Offset in feet units or meters in metric mode. The Vertical Offset normally should be set as a negative number. The bottom of the subgrade then either moves away from or towards the center depending in the Direction In or Out setting. The distance to move is specified under Distance. The Slope Type for the subgrade bottom can be either set to a specified slope or set to match the grades of the surface. After moving the specified distance, the subgrade will tie back into the template surface either by going straight up, by continuing at the subgrade slope until intersecting the surface or by wrapping around. The commonly used "continue slope" approach will extend the slope until it hits something (like a curb or another surface segment). It will not trim. So if the pavement segment is 12 feet to a curb, it is better to enter 10 and "continue slope" than to enter 12 exactly, as a "tilted" curb may place the curb edge at 11.98' from the start of the subgrade, causing the subgrade to go past face of curb and intersect back of curb. Also, for angled tie-ins of subgrade from base of curb to the surface, such as the example shown below, be sure the distance entered is less than what would intersect the surface, so that the "extend" effect will create the intersect. In this example, the first subgrade (asphalt) is "continue slope", the second (gravel) is "straight up" and the third (gravel tie in behind curb) is "continue slope".



The Material field is an optional description that is used in the *Process Road Design* report. Special super elevation pivot points may optionally be specified. The values for Horizontal Offset, Distance and Pivot Offset can be specified by template ID. For example, EP could be used in Distance to have the subgrade have a width of the EP grade. Also expressions can be used such as EP+5 to go the distance of the EP segment plus 5. This is especially useful for template transitions so that if the EP grade varies the subgrade width will automatically adjust.



Example of Wrap Around Subgrade

Keyboard Command: template
Prerequisite: None

Draw Typical Template

Function

This command draws a template and labels the slopes and distances. The cut and fill treatmentcan be shown on the left and/or right sides. All the cut/fill slopes are shown for the differentdepths when multiple slopes are defined. There are options to draw the normal template, superelevation or details of different sections.

You will be prompted to select the template (.TPL) file first, then the Typical Section dialogappears. Specify the parameters and press the Draw button.

Prompts

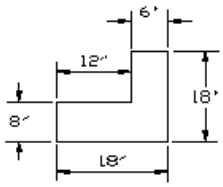
Template File to Read *Specify a template file*

Typical Section dialog *Set your options then click Draw*

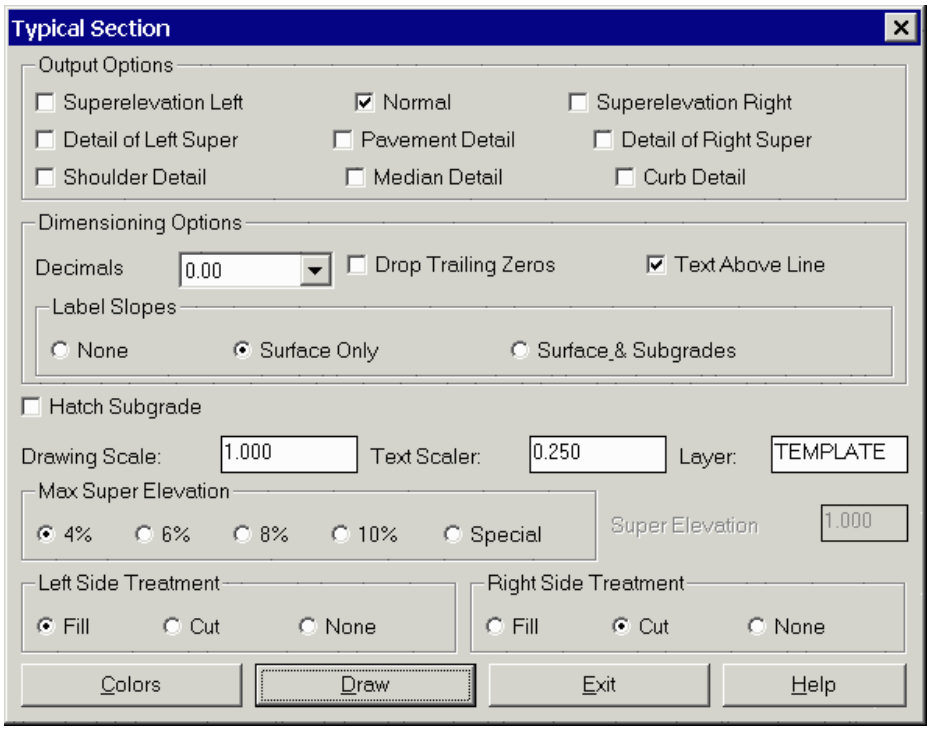
Pick Starting Position: *pick a point*

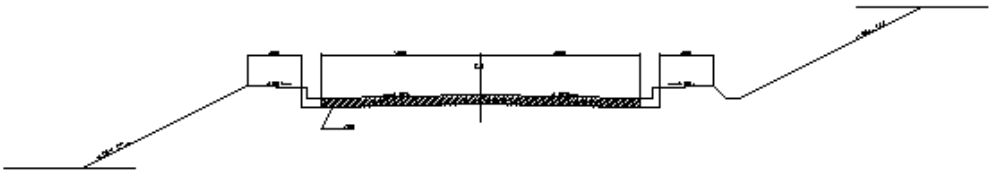
Prerequisite: A template file (.tpl file)

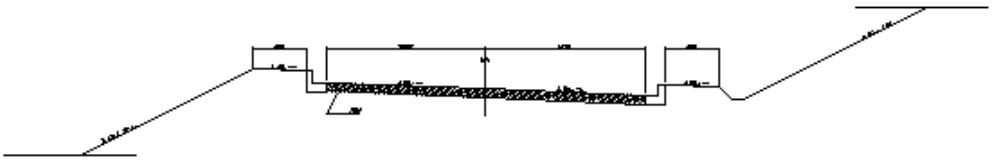
Keyboard Command: typical



Curb Detail


Typical Section
Output Options
Superelevation Left
Normal
Superelevation Right
Detail of Left Super
Pavement Detail
Detail of Right Super
Shoulder Detail
Median Detail
Curb Detail
Dimensioning Options
Decimals
0.00
Drop Trailing Zeros
Text Above Line
Label Slopes
None
Surface Only
Surface & Subgrades
Hatch Subgrade
Drawing Scale:
1.000
Text Scaler:
0.250
Layer:
TEMPLATE
Max Super Elevation
4%
6%
8%
10%
Special
Super Elevation
1.000
Left Side Treatment
Fill
Cut
None
Right Side Treatment
Fill
Cut
None
Colors
Draw
Exit
Help





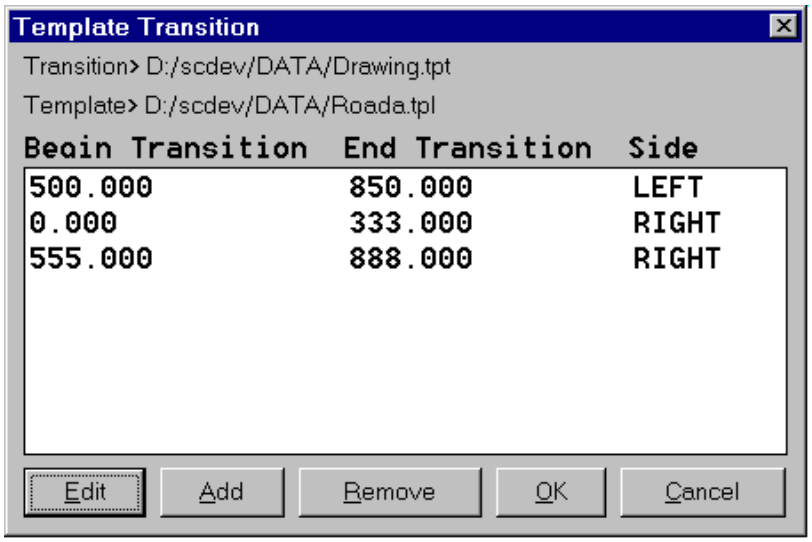
Normal Typical Template
Typical Template with Left Super Elevation

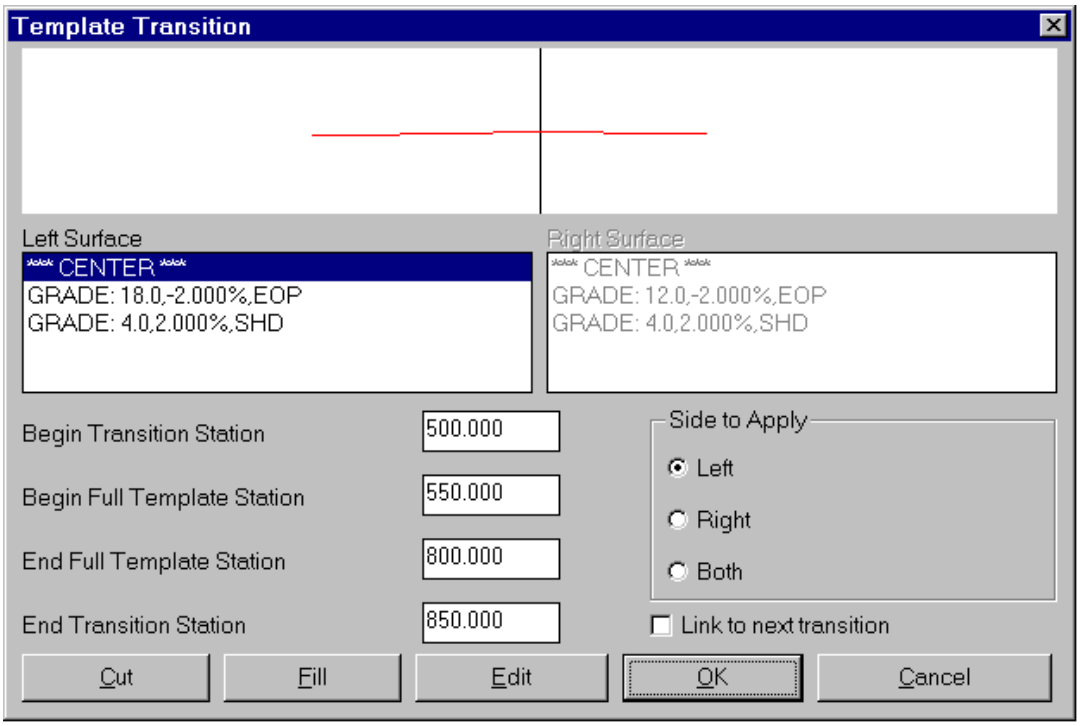
Template Transition

Function

This command creates a template transition file (.TPT file) that can be used for the commands *Locate Template Points* and *Process Road Design*. The template transition is associated with a typical template (.TPL) file. The template transition file defines changes in grade distances or slopes during a specified range of stations. The types of transitions is limited to only modifying existing template grades. Template Transition does not allow curbs, medians, subgrades or cut/fill treatment to be modified. Also new template elements cannot be added and existing elements cannot be removed.

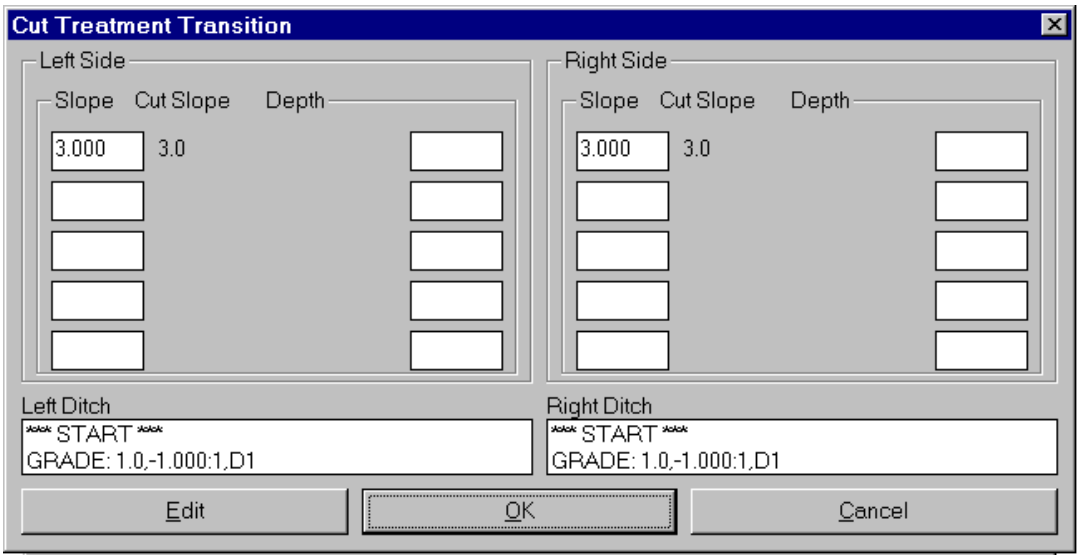
The first Template Transition dialog shows a list of the transitions. To add a transition, click the Add button. This brings up the second Template Transition dialog which shows the transition template at the top. The middle sections list the template grades that can be changed. To modify a grade, highlight the grade and click the Edit button. For example to change the EP grade from 12 feet to 18 feet,





highlight EP, click Edit and change the distance to 18. The Begin Transition Station is where the normal template begins to transition to the modified template. The Begin Full Template Station is where the modified template is used entirely. The End Full Template Station is where the template starts to transition back to normal. The End Transition Station is where the template has returned to normal. Consider the stations shown below with a normal 12' EOP grade and a modified 18' EOP grade. At station 500 EOP would be 12 feet. Between 500 and 550 the EOP distance is interpolated from 12 to 18. So at 525 EOP would be 15 feet. From 550 to 800 EOP would be

18 feet. From 800 to 850, EOP transitions from 18 back to 12 feet.



Cut and Fill slopes can also be transitioned by picking the Cut and Fill buttons. Ditch and Berm grades can also be modified here.

Transitions can also be applied to the left, right or both sides. This allows you to have separate overlapping transitions for the left and right sides.

The Link to Next Transition option joins the current transition to the next transition without returning to the normal template. For example, consider a normal template with an EOP width of 12 feet. Then this template transitions starting at station 100 to 18 feet at station 200. Then starting at station 500, the template widens to 24 feet at station 600. Next the begins transitioning back to normal at station 800 and reaches normal 12 feet at station 900. For this dual transition, make the following two transitions:

Transition #1: Set EOP to 18 feet

Begin Transition 100

Begin Full 200

Link to next transition ON

Transition #2: Set EOP to 24 feet

Begin Transition 500

Begin Full 600

End Full 800

End Transition 900

Link to next transition OFF

Prompts

New or Existing Choose New to create a transition file or Edit to modify a transition file

Template File to Edit Specify a transition file

Template Transition dialog

Prerequisite: A template file

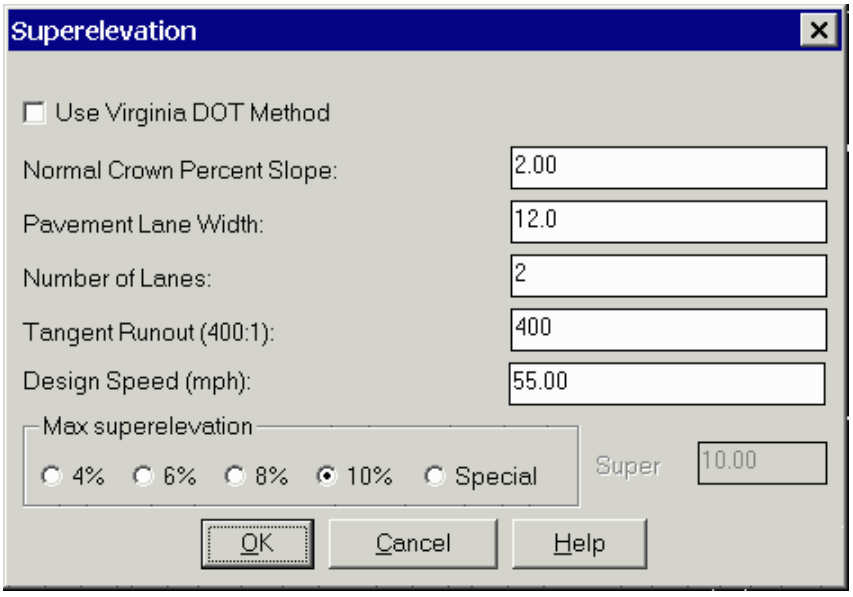
Keyboard Command: tpltrans

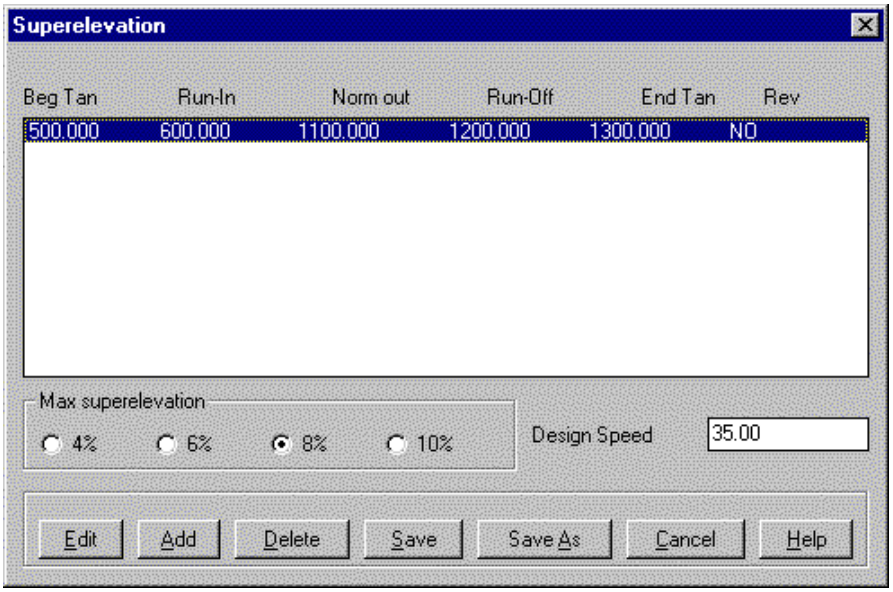
Input-Edit Super Elevation

Function

This command is an editor for super elevation stationing. The super elevation data is stored in new or existing super elevation files. These files have a .sup extension. When creating a new super elevation file, there is an option to read a centerline file and build the super elevation stationing based on the curves and spirals in the centerline by using either the Aashto-based stationing or the Virginia DOT method.

The main superelevation dialog displays a list of each super elevation transition. These entries should be sequentially entered from lowest to highest stations. To edit the super elevation stationing, highlight the entry line and click Edit. The Add button creates a new entry below the current highlighted row or at the top of the list if no row is highlighted. The Delete button removes the highlighted row from the list. The Save button saves the super elevation file. To exit the program without saving, click the Cancel button.





The super elevation stationing is entered in the Input/Edit Superelevation dialog. The View Table button shows a table of the super elevation slope for the delta angle and radius at different design speeds. The Calc Super button calculates the slope of full super given the design speed. The station entries are defined as follows:

Station to begin transition: where normal crown rate begins to transition

Station to begin super run-in: where slope becomes flat

Station for super at normal crown rate in: where slope equals negative of normal crown rate

Station to begin full super: where slope reaches full super slope

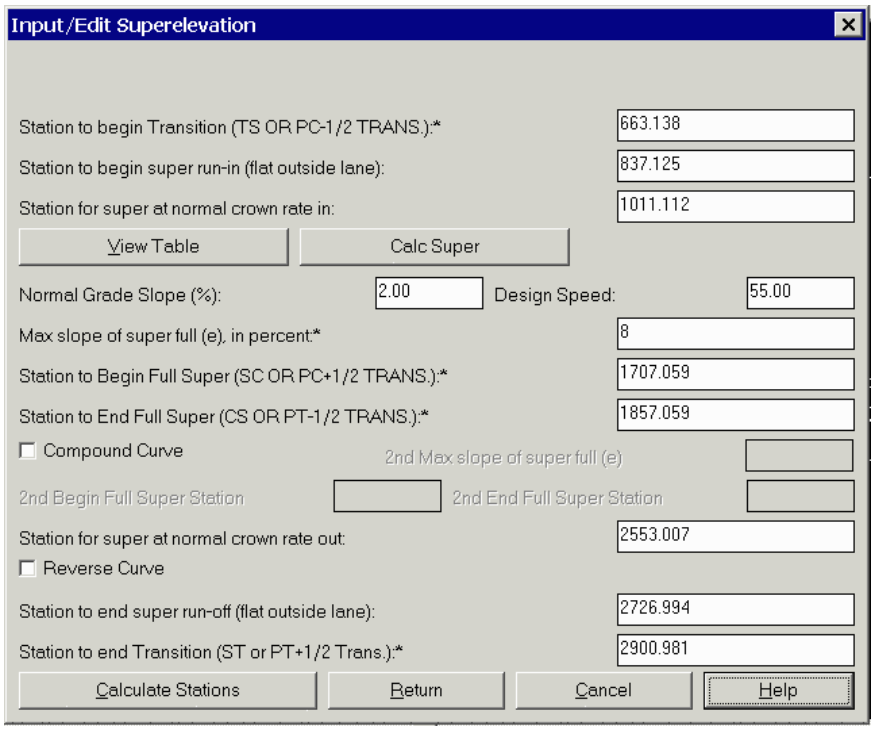
Station to end full super: where slopes begins to transition from full super back to normal

Station for super at normal crown rate out: where slope equals negative of normal crown Station to end super runoff: where slope becomes flat

Station to end transition: where slope returns to normal crown rate

The Calculate Stations button calculates the stations for begin run-in, normal crown rate in, normal crown rate out and end super run-out. To calculate these stations the values with an "*" must be entered. These values are the begin transition, begin full super, end full super, end transition, normal grade slope and slope of super full must be entered.

The Compound Curve option allows you to specify a second superelevation slope for a compound curve. In addition to specifying the second slope, the starting and ending stations for this slope must also be entered.



Prerequisite: none

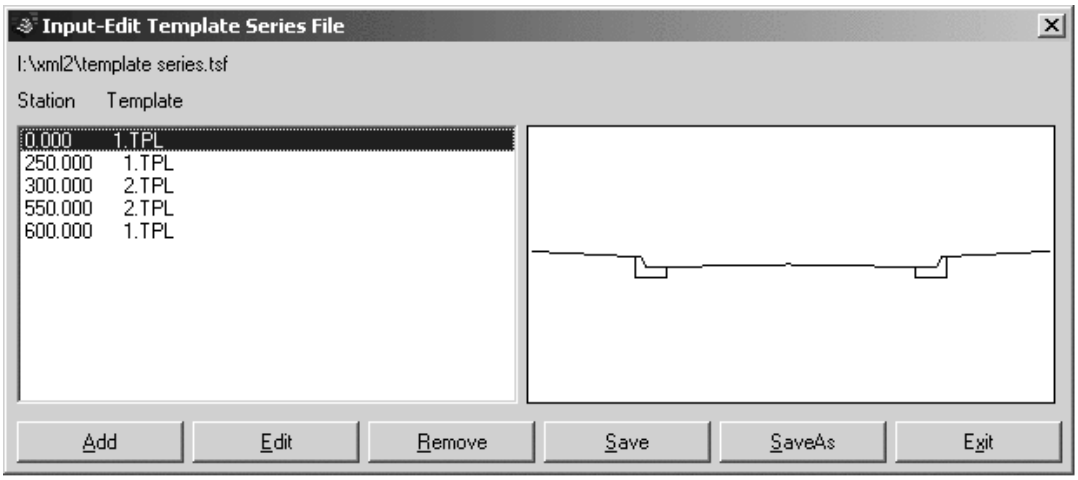
Keyboard Command: super

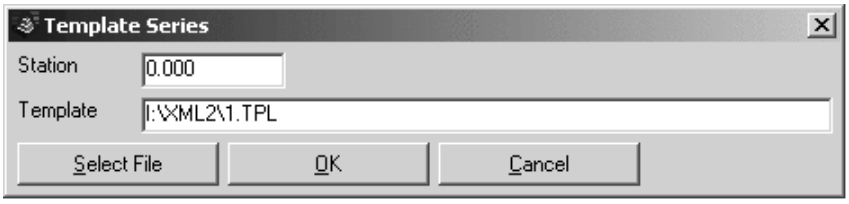
Input-Edit Template Series

Function

This routine provides the ability to simultaneously process multiple, unique templates for a single alignment in Process Road Design. Templates must first be defined with "Design Template". Templates are added to the series by selecting the desired pre-defined template files (*.TPL) and assigning a station for the beginning and ending of each template. Gaps in station ranges will create a transition between the templates. For best results, the various templates should have matching ID points for template elements. This will insure the creation of continuous template polylines when using Process Road Design.

The template series file can also be used in conjunction with Locate Template Points.





Options

Add	Imports template files and allows for station definitions.
Edit	Allows user the modify a station for an existing template in the list.
Remove	Removes a template file from the series.
Save	Writes the current template series info to a .TSF file.
Saveas	Saves the current template series info to new name.
Exit	Exits the routine.
Keyboard Command:	TPLSERIES
Prerequisite:	Two or more pre-defined template files.

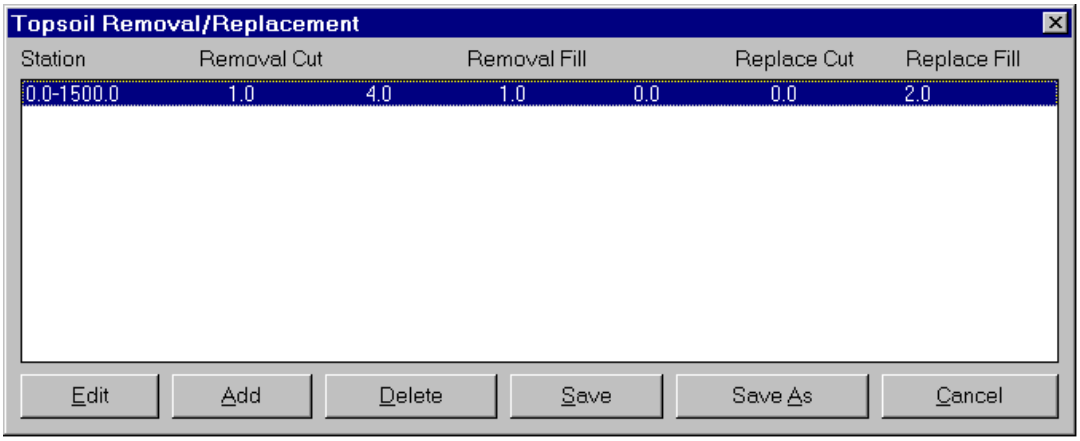
Topsoil Removal/Replacement

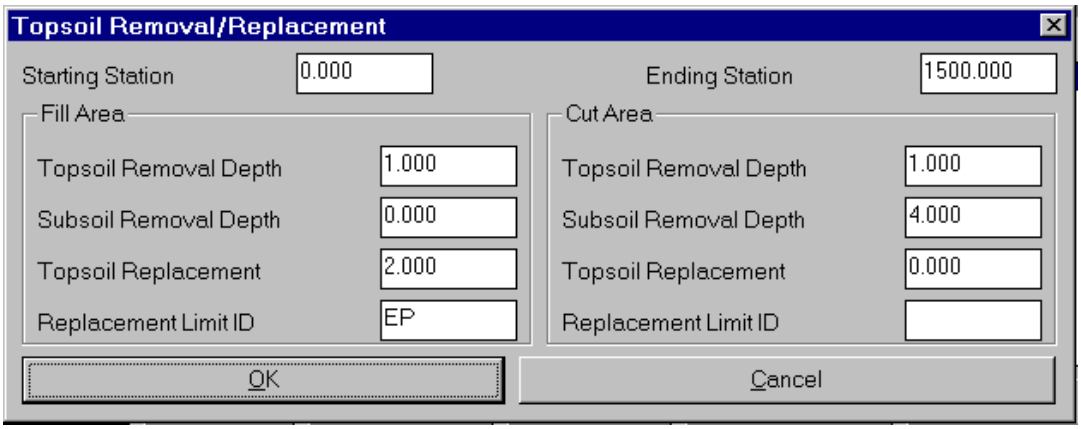
Function

This command creates a topsoil definition (.TOP) file which defines topsoil removal and replacement zones to be used in the *Process Road Design* command. You can have different topsoil adjustments for different station ranges. These adjustments are applied to the existing ground section in the *Process Road Design* command and will effect the cut and fill volumes. *Process Road Design* will also report the amounts of topsoil removal and replacement.

The command starts by displaying a list of the topsoil stations in the dialog shown below. To add a topsoil adjustment, pick the Add button which brings up a second dialog. You can have different amounts of topsoil removal and replacement for areas in cut and areas in fill. Subsoil is another category of removal that will be combined with any topsoil removal. The Subsoil removal volume is reported separately from topsoil removal by Process Road Design.

The Replacement Limit ID is an option to limit the replacement to occur only within the template left offset Limit ID and the right





offset Limit ID. If this Limit ID is left blank, then the program will apply the replacement between the left catch point and the right catch point. Topsoil removal is always applied between the catch points. The Limit ID corresponds to a template ID as set in the Design Template routine.

Prompts

Topsoil File to Read Specify a topsoil file

Topsoil dialog *Choose your options*

Prerequisite: None

Keyboard Command: topsoil

Assign Template Point Profile

Function

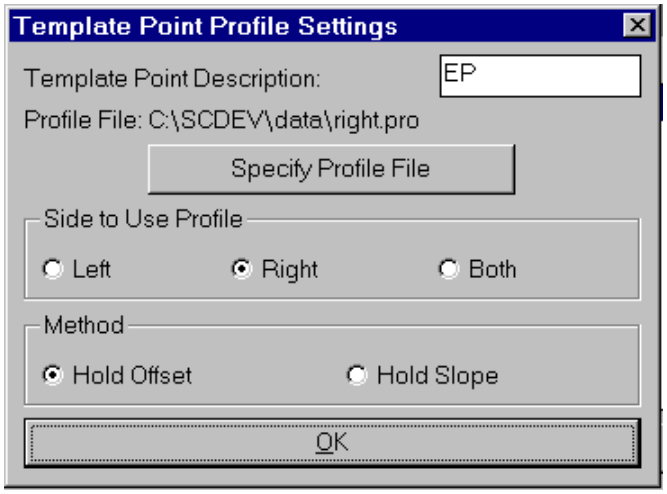
This command assigns profile (.PRO) files to template points, storing this information in a template point profile (.TPP) file which is used by the command *Process Road Design*. The purpose of the profile assignments is to allow separate profiles for template points that are independent of the centerline profile. For example, a ditch grade could have a different profile than the centerline. Multiple template point profiles can be assigned so that amount of control is unlimited. The Template Point Description corresponds to the name set in the *Design Template* command.

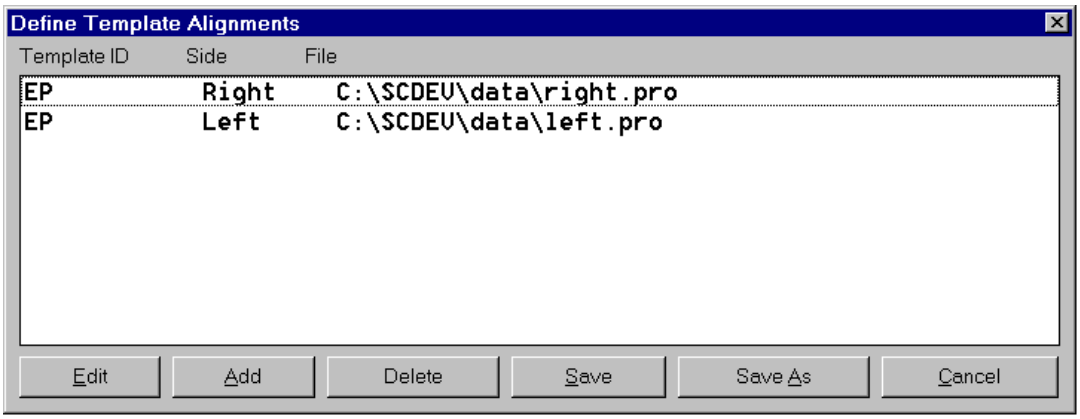
The command starts by showing a list of the profile assignments. To add a new profile, pick the Add button. This brings up another dialog where you enter the Template Point Description. To set the profile, choose the Specify Profile File button. For example, to have the ditch grade follow an independent profile, choose the ditch profile (.PRO) file and enter the template ID for the ditch grade in the dialog shown.

Since the template ID profile can change the relative position of the template ID from the centerline, you have two options for how to fit in the template ID profile: Hold Offset or Hold Slope. Hold Offset will keep the same offset for the template ID and adjust the slope to the template ID. The Hold Slope will keep the same slope to the template ID and adjust the offset to reach the template ID profile elevation.

Prerequisite: Profile file (.pro)

Keyboard Command: tppset





Assign Template Point Centerline

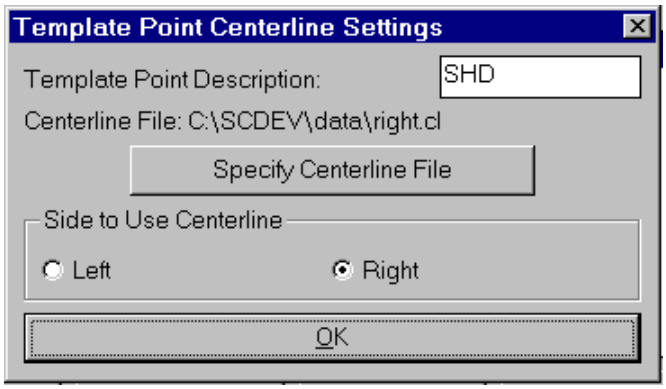
Function

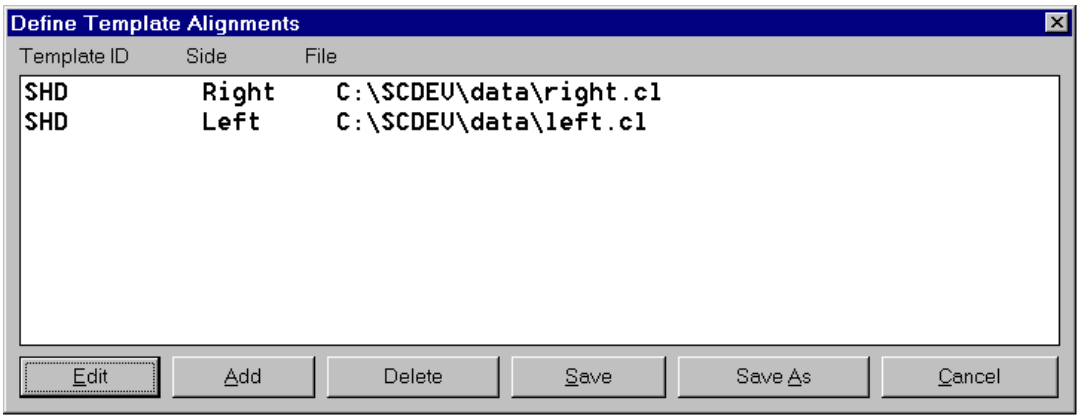
This command assigns centerline (.CL) files to template points, storing this information in a template point profile (.TPC) file which is used by the command *Process Road Design*. The purpose of the centerline assignments is to allow separate centerlines for template points that are independent of the main centerline. The offsets of the template points will follow their assigned centerline. The slope to these template points will use the template slope defined in *Design Template*. For example, if the template shoulder point must follow an existing road shoulder which has offsets that vary from the centerline, then you could assign a Template Point Centerline to this shoulder point.

The command starts by showing a list of the centerline assignments. To add a new centerline, pick the Add button. This brings up another dialog where you enter the Template Point Description which corresponds to the name set in the *Design Template* command. To set the centerline, choose the Specify Centerline File button.

Prerequisite: Centerline file (.cl)

Keyboard Command: tpcset





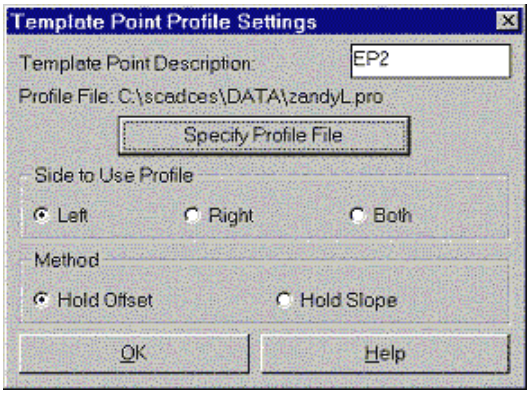
Using Template Point Profiles and Template Point Centerlines

We have a road that has the EP (edge of pavement) move around on the left side to follow an existing curb. Assume the road is 800 feet long. The first 12 feet (CL to EP) will remain unchanged, but the "lane" beyond 12 feet to EP2 will vary in width and slope. To follow existing, we assign a profile and a centerline to the EP2 template point. (See below). We make a profile for EP2 stationed 0 through 800. The EP2 centerline (because it is not straight and parallel to the main line) is 0 through 801.16. That is normal. The profile should be entered 0 through 800 for the EP2 point-it should reference the road centerline. The program automatically matches 800 on the main road template to 801.16 on the EP2 template point by cutting a cross section to obtain offset distance. So the centerline and EP2 profiles are entered from 0 to 800 and must match, but the EP2 centerline will naturally have different (longer) stationing.

Within *Assign Template Point Profile*, use Hold Offset (the profile should not "move" the offset). See here:

Make the EP2 position (a left-lane running

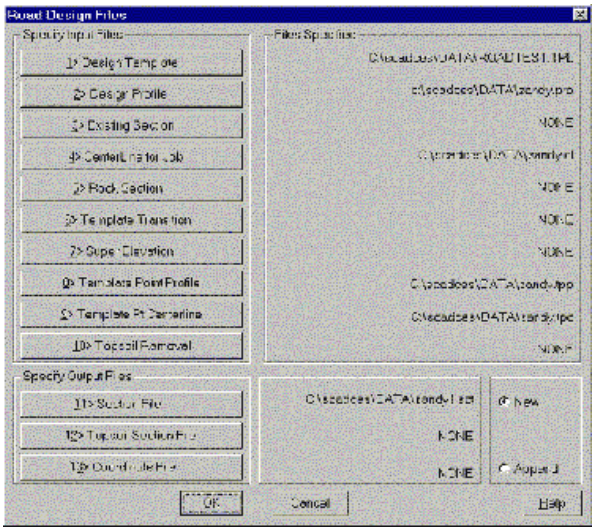
EP2 ends at station 801.16
But c/l ends at 800

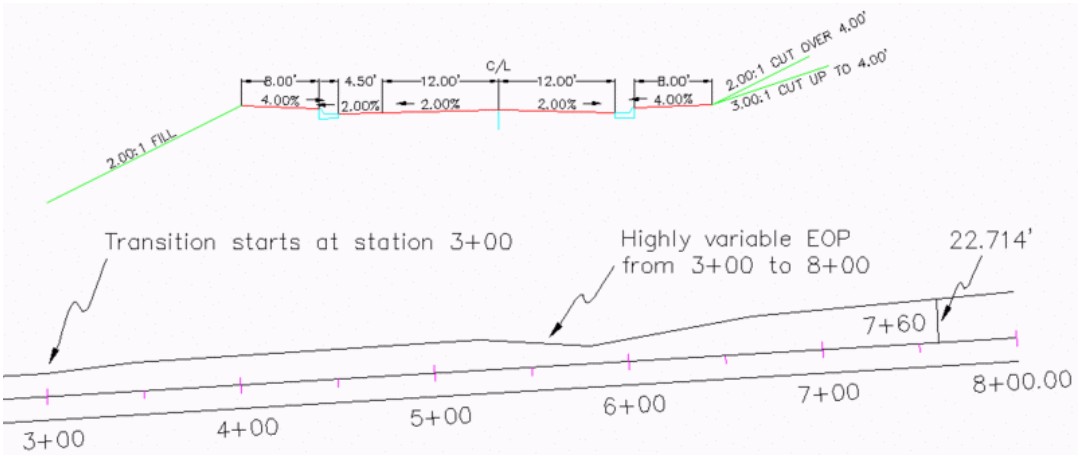


beyond the first lane) follow the zandyl.pro for vertical and the zandyl.cl (which is the upper, irregular centerline going to 801.16 rather than 800). When you run this out and process the road design, it correctly follows the profile (dropping down an extra _0.24 at station 800, for example). And it correctly moves out to the cl position

Here is the dialog leading into the result:

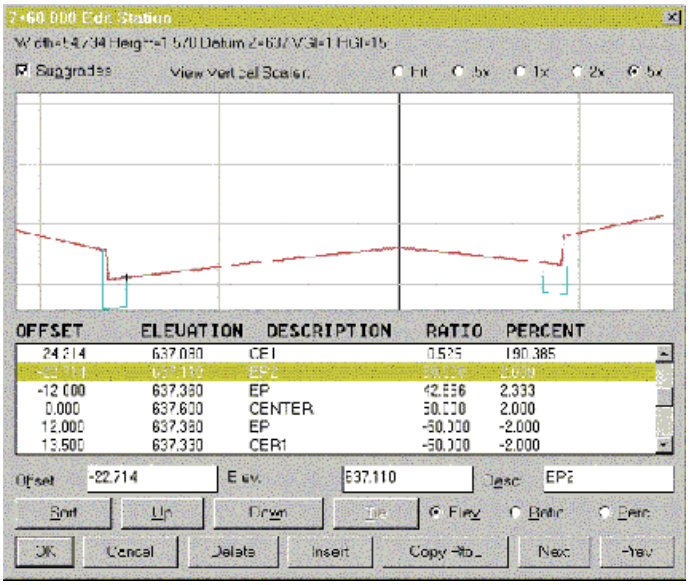
Here is the drawing, in close-up view:

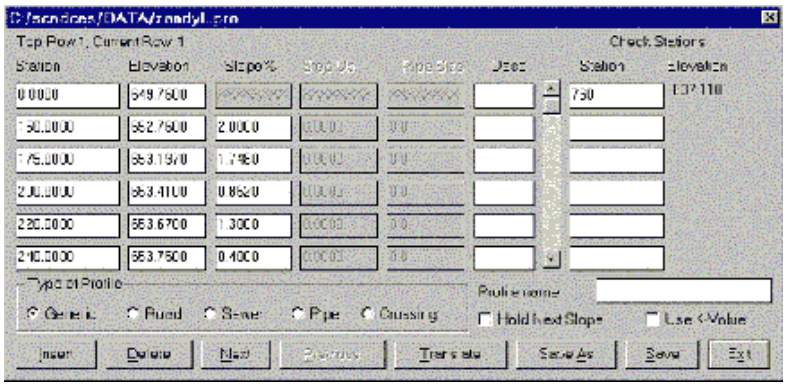




We have used AutoCAD to compute the distance from centerline to EOP at station 7+60, and the distance is 22.71. This same distance is computed by the program, as confirmed in the following dialog from Input-Edit Section:

Note that the profile elevation is 637.110. This profile elevation can also be confirmed by using the Check Station option within Input-Edit Profile, for Station 7+60 on ZandyL.pro. This confirms that the system is working as designed.



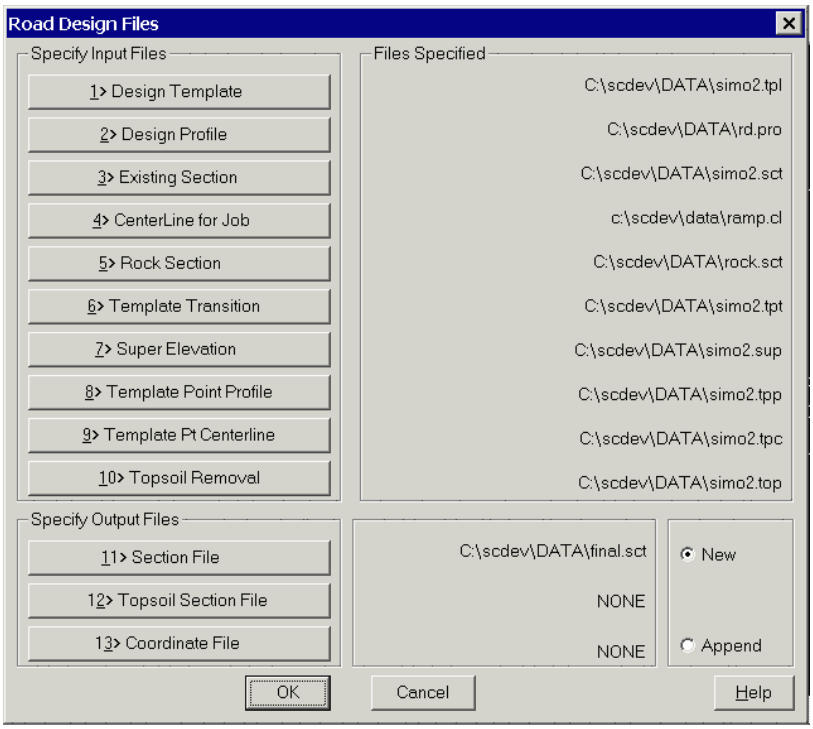


Process Road Design

Function

The primary function of this command is to apply the design template at the design profile elevation and compute the outslopes and earthworks relative to the existing ground section file.The earthworks report is shown in the standard report viewer. Secondary functions include creating a final grade section file for plotting with the Draw Section File command, creating final grade points in a coordinate file, creating a final surface/contour model, and drawing the road as 3D polylines. The program also has options for defining a super elevation file, template transition file, template point profile, template point centerline, rock section file and a centerline file. All the input design files must be created before starting this command.

This command begins with the dialog shown below. The top section contains input design files. As a minimum you must specify file names for Design Template and Design Profile. With just these two files, the program will run the template with the elevations of the profile vertical alignment which can create final sections and final coordinates. To calculate earthworks you must specify an Existing Section file. In order to locate the template in the drawing, a centerline file must be supplied. With a centerline, the template can be drawn as 3D polylines and final contours can be generated.



The other files are optional design files.

The Output Files section allows you to specify files to store the processing results. The Section File creates a final grade section file that can be drawn with Draw Section File. The Topsoil Section File creates the modified existing ground section file if Topsoil Removal is set in the input.The Coordinate File creates a coordinate file containing every break point in the final grade. The point descriptions include the station, offset and template ID. Whether to include the subgrade points as well as the final surface points is determined by the Include SubGrade Points in Output CRD File option. To the right of the Output Files is the option to create new output files or append to existing output files. For example, you could have one template for stations 0-1000 and another template for stations 1000-2000, and you want to store the results in one set of output files. The first time that you run this command for stations 0-1000, you would set Output Files to New. Then you could run this command again with the second template for stations 1000-2000 and set Output Files to Append.

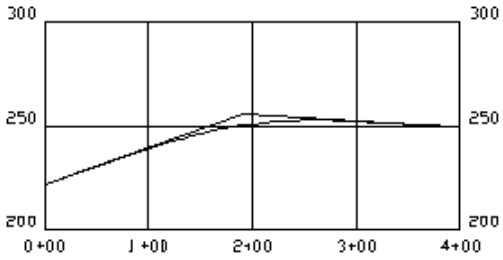
1 Design Template

Specify a template definition (.TPL) file that defines the final grade offsets and elevations and the cut/fill slopes. This file is created by the Design Template command and is always required to run Process Road Design.

2 Design Profile

Specify the design profile (.PRO) file to derive the centerline elevations from when the template is applied. This file defines the vertical alignment and is always required. The profile can be created with any of the profile creation routines in the ∗Profile∗ menu, but typically you would use Design Road Profile.

Example Design Template



Example Design Profile

3 Existing SeCTion

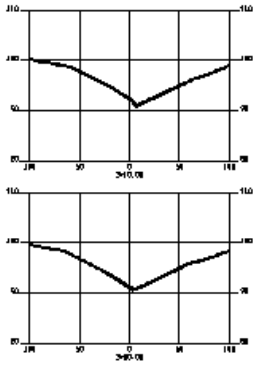
Specify the section (.SCT) file which will be treated asthe existing ground for cut and fill volumes and to calculate the outslope intersections when the template is applied at profile centerline elevations. This file can be created with commands such as Sections from Surface Entities, Input/Edit Section File, Sections from Points or one of the Digitize Sections commands on the Section menu.

4 Centerline for Job

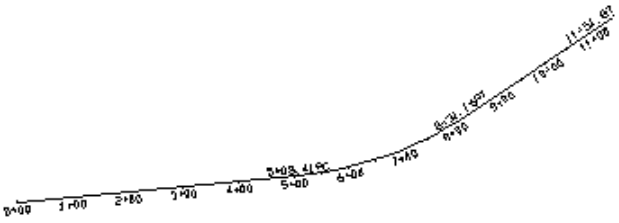
Specify the name of the CenterLine file with this option. The .CL file contains the horizontal alignment geometry for a project. This parameter file must be specified if you want to have earthworks centroid corrections computed, generate Output Files (options 9 & 10), Disturbed Area Polyline, and/or use Triangulate & Contour. The centerline file can be created by the Design Centerline or Polyline to Centerline commands in the Roads pull-down menu.

5 Rock Section File

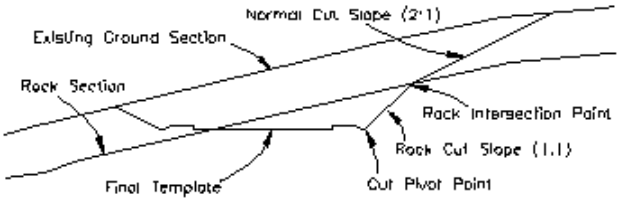
This option specifies an optional rock section file that is used as an additional surface. When in cut, a special cut slope is used up to the intersection of the rock surface. After this intersection, the normal cut slopes apply. The special rock cut slope is specified in Design Template under the cut options.



Example Existing Sections



Example Centerline



Detail of rock cut slope

6 Template Transition File

Specify a .TPT file with this option. The Template Transition file allows modified template files to be applied at different ranges of stations on a project. Use the Template Transition command under the Design menu to create a template transition file.

7 Super Elevation File

This option is used to specify a super elevation file (.sup file) that defines the superelevation transition stations on a project. The super elevation file can be created with the Input-Edit Super Elevation command.

8 Template Point Profile

This option lets you have separate profiles for template points that are independent of the centerline profile. This design file is created with the Assign Template Point Profile command.

9 Template Point Centerline

This option lets you have separate centerlines for template points that are independent of the main centerline. This design file is created with the Assign Template Point Centerline command.

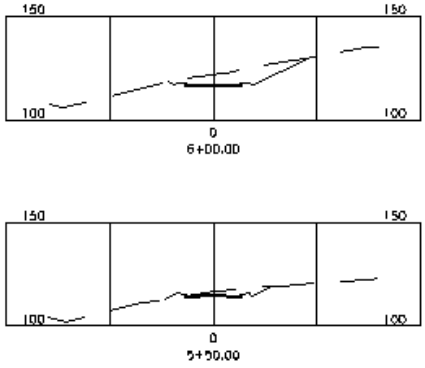
10 Topsoil Removal

This option applies topsoil removal and/or replacement to the existing ground section file. This design file is create with the Topsoil Removal/Replacement command.

11 Output Section File

Specify the name of the file to output the final grade sections calculated by applying the template file at profile elevations and calculating the outslope intersection withthe existing ground cross sections. This file can then be plotted by using the Draw SeCTion File command. After plotting the final sections overlaidon the existing sections, revisions can be made graphically with commands like PEDIT and Polyline by SlopeRatio. The data output to the file can also be edited and reviewed with the Input-Edit SeCTion File command. If the final sections are edited graphically, the revised section data can be updated in the .SCT file with the Polyline to SeCTion File command.

12 Output Topsoil File



Output Section File draw with Existing Section File by the Draw Section File command

This option writes out the modified existing ground section to the specified section file name. This option is only valid if a Topsoil Removal file is being used.

13 Output Coordinate File

This option creates a coordinate file containing every break point in the final grade for the range of processed stations. The point descriptions include the station, offset and template ID. The station interval is set by the stations in the Existing Section File.

Running the Road Design Job

After setting up the files and options in the first dialog click the OK button. The next dialog shown below has processing options.

Input the range of station that you want to calculate in the first edit box. Each time you use this command the existing grade .SCT file is scanned and the range in the edit box is set to the minimum and maximum stations in the file.

Volumes are calculated using end areas between the range of stations. Instead of cutting off the volumes exactly at this range, the Ending and Starting Stations for Cut and Fill can be used to have the volume taper from zero at the specified Starting Station to the volume at the first station inthe range. Likewise the Ending Stations can be used to taper the volume from the last station in the range to zero at the specified Ending Station.

The Shrink and Swell Factor edit boxes allow you to specify a value that the volume calculated will be multiplied by. If you specify any number other than one an additional report will be sent to the printer device showing accumulated adjusted volumes and differences.

The Vertical Offset from Profile of Design Profile edit box allows you to specify a distance that
PtNo. North(y) East(x) Elev(z) Description

122 189497.42 611730.32 90.01 TIE 0+90.00L53.65

123 189461.43 611733.72 108.09 SHD 0+90.00L17.50

124 189457.45 611734.09 107.93 CURB3 0+90.00L13.50

125 189456.95 611734.14 107.93 CURB2 0+90.00L13.00

126 189456.95 611734.14 107.09 CURB1 0+90.00L13.00

127 189455.96 611734.23 107.09 EP 0+90.00L12.00

128 189444.01 611735.36 107.33 CENTER 0+90.00R0.00

129 189432.06 611736.49 107.09 EP 0+90.00R12.00

130 189431.07 611736.58 107.09 CURB1 0+90.00R13.00

131 189431.07 611736.58 107.93 CURB2 0+90.00R13.00

132 189430.57 611736.63 107.93 CURB3 0+90.00R13.50

133 189426.59 611737.00 108.09 SHD 0+90.00R17.50

134 189412.18 611738.36 100.85 TIE 0+90.00R31.97
Example coordinates for station 0+90.

will be added to the profile elevation when the template is placed at profile grade.

The Horizontal Offset of Template will offset the template design from the centerline. Use apositive value for to offset to the right and use a negative value to offset left.

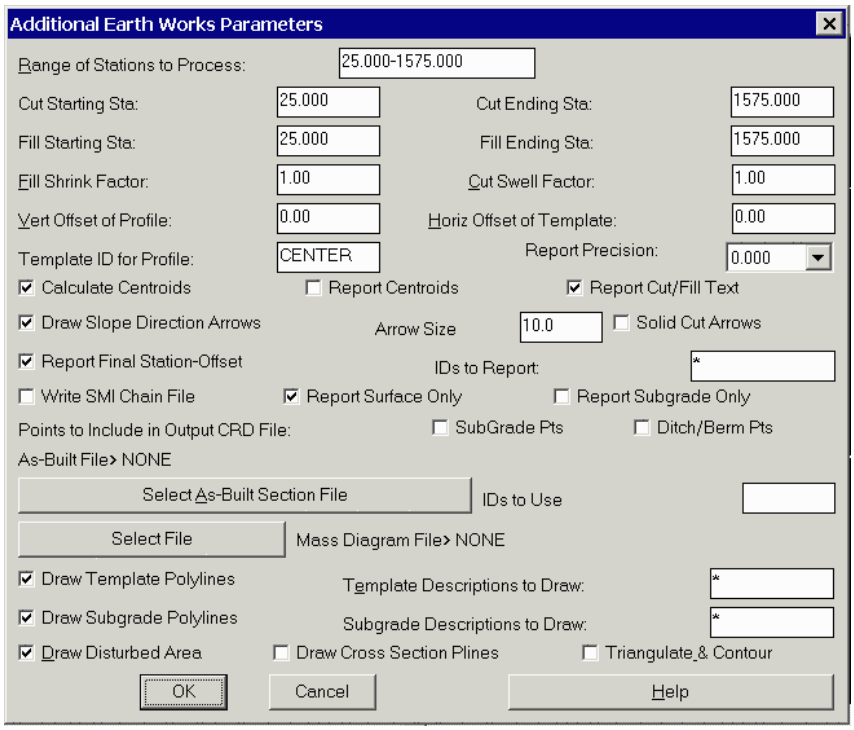
The Template ID for Profile identifies the template grade ID that applies to the design profile. The default of CENTER is to have the profile apply to the template center. This option allows youto use a profile for a template point other than the center such as a profile for the edge ofpavement. In this case, if the design template file has an ID of EP for the edge of pavement, then enter EP for this option.

Write SMI Chain File creates a chain (.CH) file that contains the centerline, profile and template data for SMI Construction V.

The Calculate Centroids option determines whether the program applies the centroid adjustment to the station interval for the volume calculations. With this option on, the program calculates the offset of the centroid for each cut/fill end area. This offset is used to calculate the station interval which can be longer on the outside of a curve and shorter on the inside of a curve.

The Report Centroids toggle controls whether the shift in the cut or fill centroid radius shiftwill be included in the earthworks report.

The Report Cut/Fill Text option will include the cut/fill for each station in the report.



The Draw Slope Direction Arrows option will draw arrow indicators for cut or fill slope direction. The arrows are drawn in plan view and usually are drawn together with the Draw Disturbed Area and Draw Cross Section Plines options. Cut arrows start from the disturbed area limit and point towards the centerline. Fill arrows start from the base of the fill slope and point away from the centerline. The Solid Cut Arrows option chooses between solid fill or wire-frame cut arrows.

The Report Final Station-Offset option will create a report of the final section offset-elevation data in row-column format. The station and profile grade are shown on the left followed by columns of offset and elevation for each data point. There are options to report the surface points only, the subgrade points only or filter the points by ID.

Select the Mass Diagram Output File button if you want to specify the name of a .PRO file to output haul data to.

The bottom section of the Additional Earthworks Parameters dialog contains output options which are only available when a ceneterline file is specified. The Draw Cross Section Polylines option will create 3D polylines perpendicular to the centerline with each template break point. The interval of these cross section polylines is determined by the station interval of the Existing Sections. The Draw Template Polylines option will create 3D polylines parallel to the centerline by connecting common template point ID's. For example, a template ID could be EP which this option woulduse to create 3D polylines for EP on the left and right of the centerline. Which template point ID's to connect in set under Template Descriptions to Plot. Setting this to an asterick (*) will plot all the template break points. The Draw Disturbed Area Polyline option will create a polyline perimeter that represents where the cut/fill slopes tie into the existing ground. The Triangulate & Contouroption will automatically run this command after Process Road Design is done to create the finalcontours. Triangulate & Contour uses the template 3D polylines to model the final surface, and the disturbed area polyline is used as the inclusion perimeter for the contours. With Triangulate & Contouractive, Draw Template Polylines and Draw Disturbed Area Polyline are automatically turned on.

Final Surface Section Report

STATION P.G.

2+50.000 1013.444 59.619 18.000 12.000 0.000 12.000

992.634 1013.444 1013.204 1013.444 1013.204

2+75.000 1015.059 65.772 18.000 12.000 0.000 12.000

991.173 1015.059 1014.819 1015.059 1014.819

3+00.000 1016.499 71.547 18.000 12.000 0.000 12.000

989.725 1016.499 1016.259 1016.499 1016.259

3+25.000 1017.764 76.733 18.000 12.000 0.000 12.000

988.398 1017.764 1017.524 1017.764 1017.524

Portion of final station-offset report

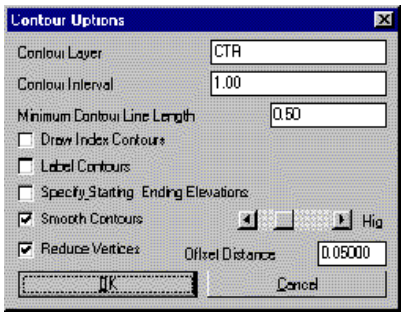
Prompts

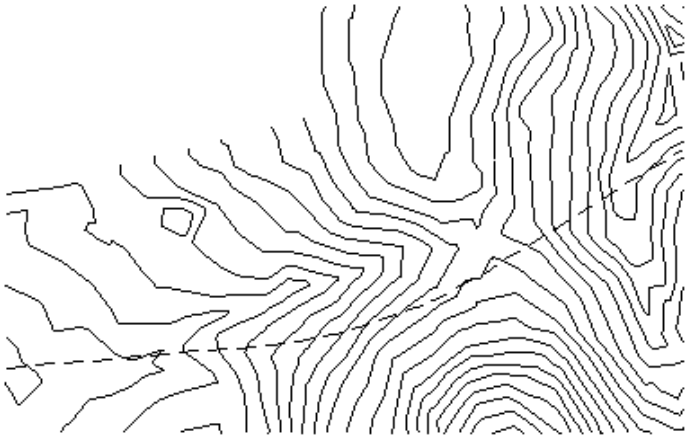
Road Design Files dialog Choose the design files
Additional Earth Works Parameters
Road Design Report dialog
Contour Options This dialog appears if Triangulate & Contour is on. Usually you would want to turn Smooth Contours off.
Trim existing contours inside disturbed area (Yes/<No>)? Y for yes. This prompt appears if Triangulate & Contour is on. This option will trim polylines with elevation that cross the disturbed area perimeter for the road.
Join final contours with existing(<Yes>/No)? Y for yes. This prompt appears if Triangulate & Contour is on. This option will join the final contours with the existing contours where they join at the disturbed area perimeter.

Keyboard Commands: eworks

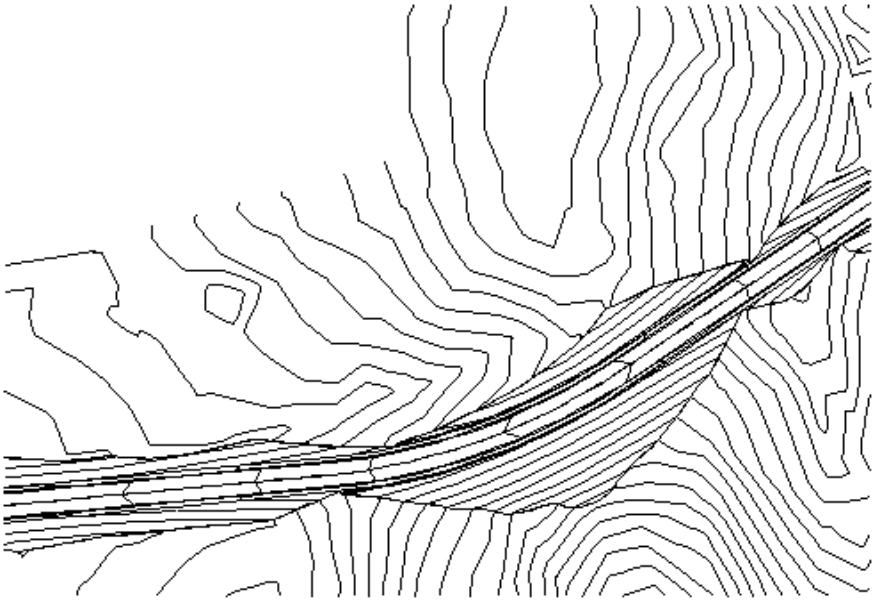
Prerequisite: Profile file and template file.

File Names: \lsp\eworkd.lsp, \lsp\eworks.arx, scadewrk.dcl
Template File> C:\scadxml\DATA\simo2.tpl
Profile File> C:\scadxml\DATA\rd.pro
Existing Section File> C:\scadxml\DATA\simo2.sct
Centerline File> C:\scadxml\DATA\simo2.cl
Processing 0+25.000 to 7+51.152
Total Cut : 800563.177 C.F., 29650.488 C.Y.
Total Fill: 1554948.266 C.F., 57590.677 C.Y.
Station Cut(sf) Fill(sf) Interval Cut(cy) Fill(cy)
0+25.000 4407.456 0.000
25.000 4784.266 0.000
0+50.000 5926.559 0.000
25.000 5535.921 0.000
0+75.000 6031.029 0.000
25.000 4840.888 0.000
1+00.000 4425.290 0.000
25.000 3432.528 0.000
1+25.000 2988.971 0.000
25.000 2713.262 3.362
1+50.000 2871.676 7.262
Portion of earthworks report

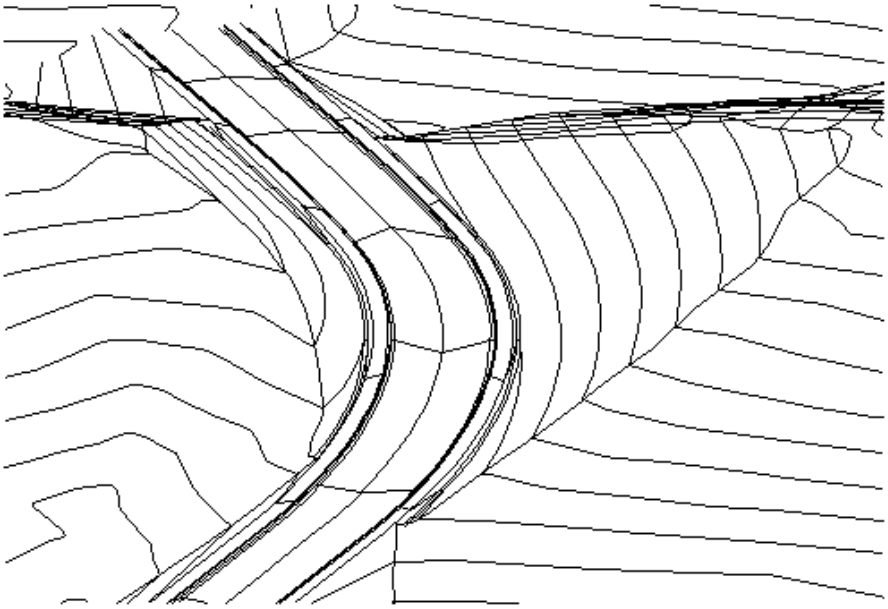




Existing Contours and Centerline



3D template polylines, disturbed area perimeter polyline and final contours



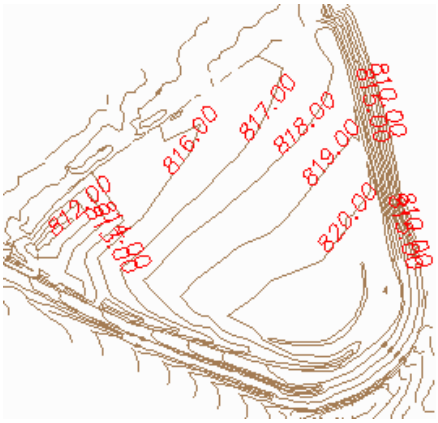
Template polylines and final contours viewed in 3D using Viewpoint 3D command

Display Menu 15

Existing Drawing

Function

This command allows you to display all the entities on the layers that are grouped as part of the Existing Drawing.



Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Existing Drawing is checked than the existing drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the existing drawing in isolation or in accordance to the other drawings.

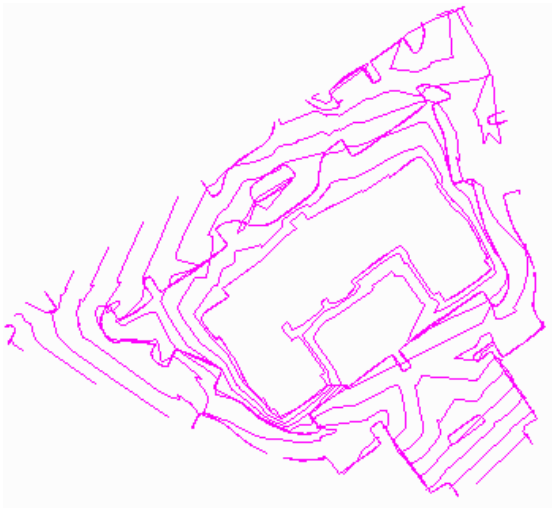
Prerequisite: Define Layer Target/Material/Subgrade

Keyboard Command: set_display_exist_dwg

Existing Contours

Function

This command displays all the contours that represent the existing surface (For contouring options see Display Options). Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



When Existing Contours is checked than all the contours for the existing surface will be displayed. If it is not checked they will not be displayed.

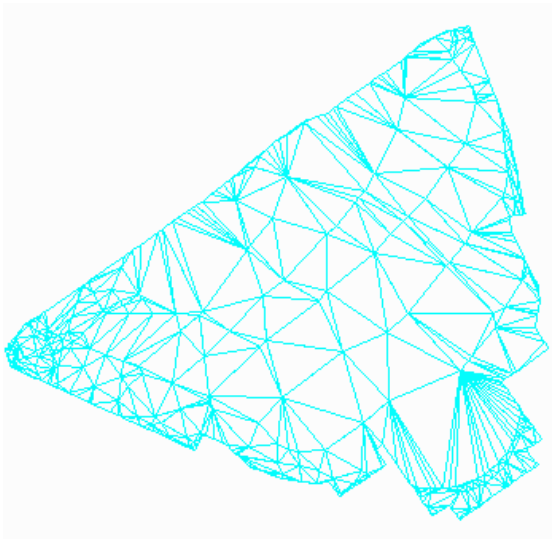
Prerequisite: existing surface

Keyboard Command: set_display_exist_ctr

Existing Surface

Function

This command allows you to display the surface triangulation for the existing drawing.



When Existing Surface is checked than all the triangulation for the existing will be displayed. If it is not checked, they will not be displayed.

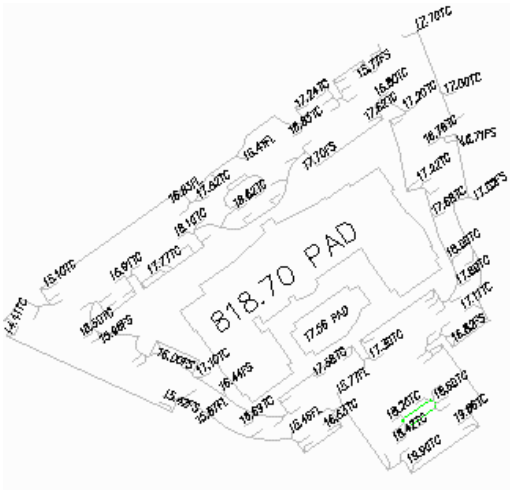
Prerequisite: an existing surface

Keyboard Command: set_display_exist_grd

Design Drawing

Function

This command allows you to display all the entitles on the layers that are grouped as part of the Design Drawing.



Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. When Design Drawing is checked than the design drawing will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Design drawing in isolation or in accordance to the other drawings.

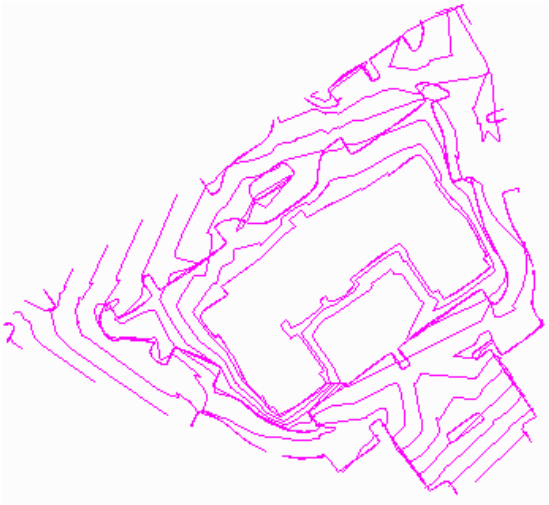
Prerequisite: Define Layer Target/Material/Subgrade

Keyboard Command: set_display_final_dwg

Design Contours

Function

This command displays all the contours that represents the design surface (For contouring options see Display Options).Clicking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



When Design Contours is checked than all the contours for the design will be displayed. If it is not checked they will not be displayed.

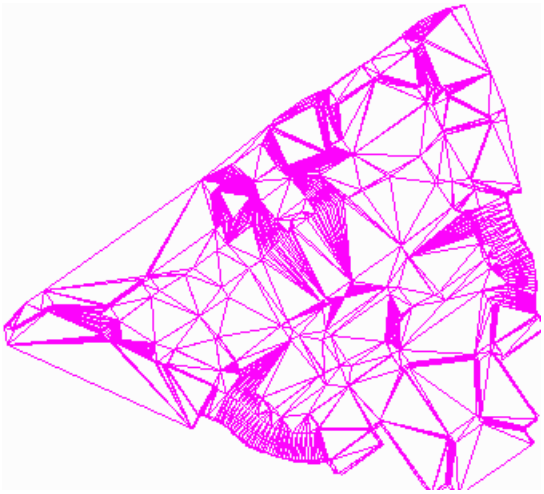
Prerequisite:design surface

Keyboard Command: set_display_final_ctr

Design Surface

Function

This command allows you to display the surface triangulation for the design drawing.



When Design Surface is checked than all the triangulation for the design will be displayed. If it is not checked, they will not be displayed.

Prerequisite: a design surface

Keyboard Command: set_display_final_grd

Cut Fill Contours

Function

This command compares the existing and design surfaces and shows the cut/fill contours in blue for fill and red for cut. There is a Draw Only Cut/Fill Daylight option as apart of the Display Options command (See Display Options for more information). Clicking on Cut/Fill Contours from the menu runs the command and puts a check mark on the menu. Picking again turns it off.



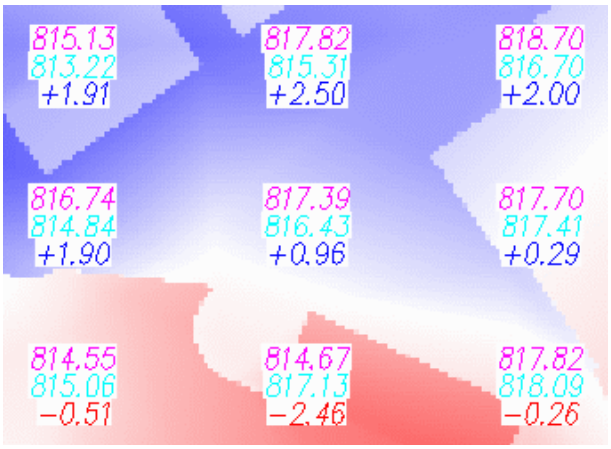
Prerequisite: elevation differences between existing and design

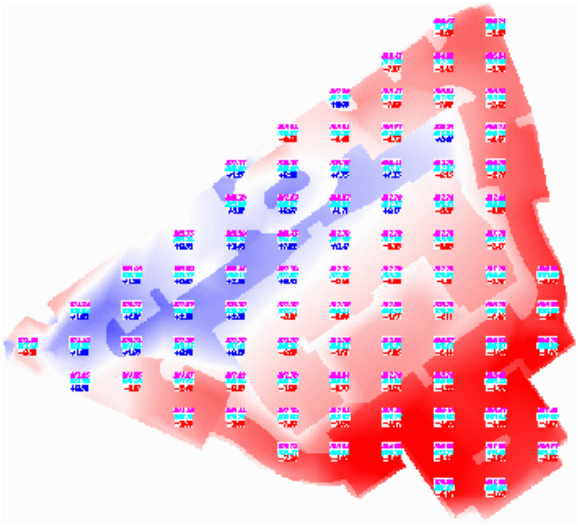
Keyboard Command: set_display_cf_ctr

Cut Fill Labels

Function

This command displays the design elevation, the existing elevation, and the amount to either cut or fill right on the screen (See Display Options for infromation about labeling options). Picking on Cut/Fill Labels from the menu runs the command and puts a check mark on the menu. Picking again turns it off.





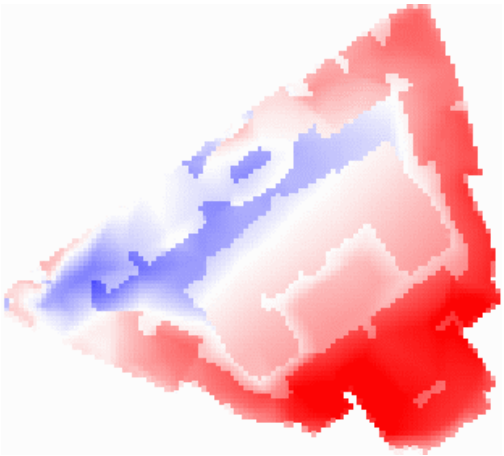
Prerequisite: existing and design surfaces

Keyboard Command: set_display_cf_txt

Cut Fill Color Map

Function

This command compares the existing and design surfaces and shows the cut/fill regions in blue for fill and red for cut (See Display Options for information on pixel resolution). Clicking on Cut/Fill Color Map from the menu runs the command and puts a check mark on the menu. Picking again turns it off.

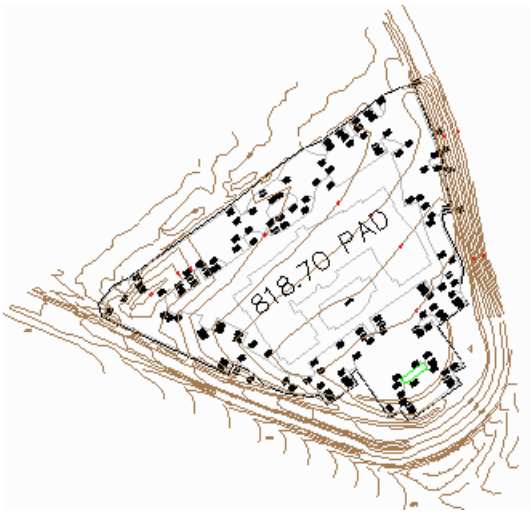


Prerequisite: existing and design surfaces
Keyboard Command: set_display_cf_map

Other Drawing

Function

This command allows you to display all the entities on the layers that are grouped as part of the Other drawing.



Carlson TakeOff allows you to assign layers into three different "Target" surface groups: Existing, Design, and Other. For more about assigning layers to different "Target" surface groups see Define Layer Target/Material/Subgrade under the tools menu. Once layers have been assigned, the display menu allows for easy viewing of each "Target" surface. Typically, most layers are listed under Other before they are assigned to Existing or Design. Some layers, like perimeter, are neither apart of the Existing or the Design drawing so they remain under Other. When

Other Drawing is checked than the entities grouped under Other will be displayed. If it is not checked it will not be displayed. You can check on and off the other "Target" surfaces to view the Other surface in isolation or in accordance to the other surfaces. In this example, Existing, Design, and Other are all shown.

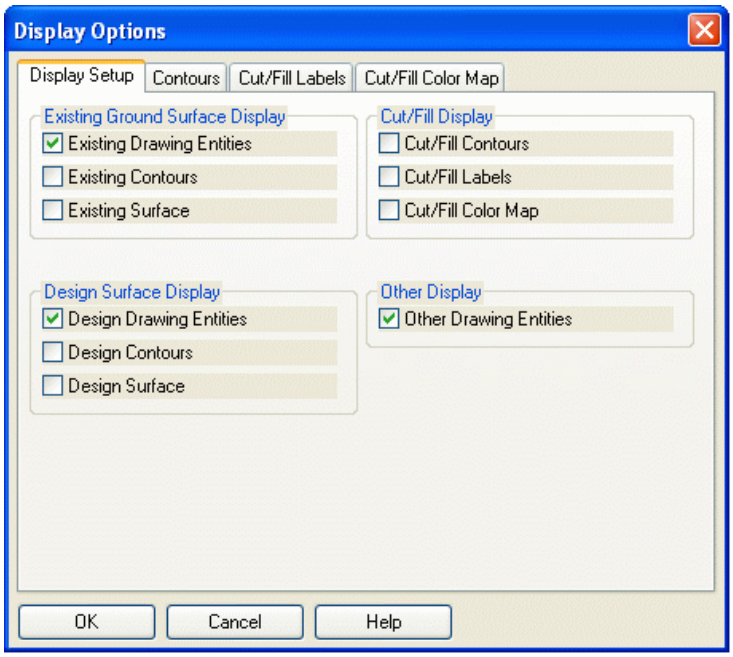
Prerequisite: Define Layer Target/Material/Subgrade

Keyboard Command: set_display_other_dwg

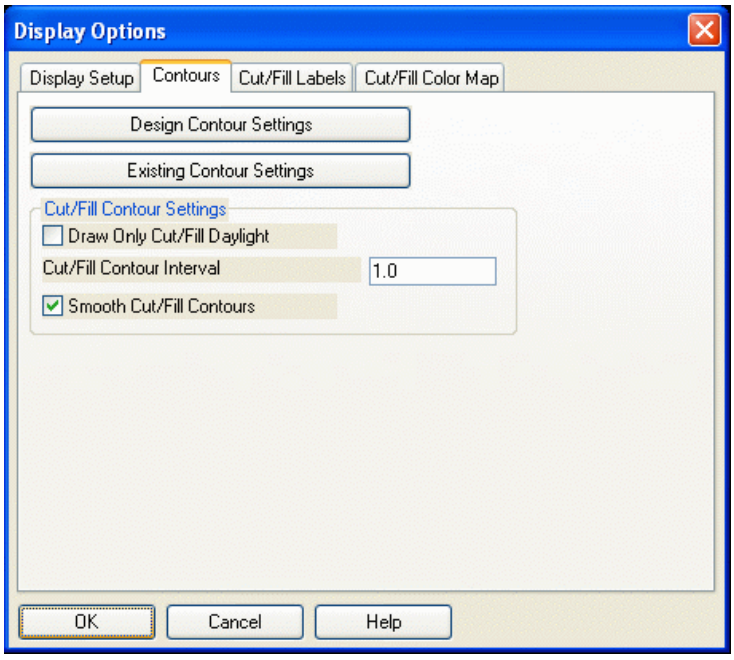
Display Options

Function

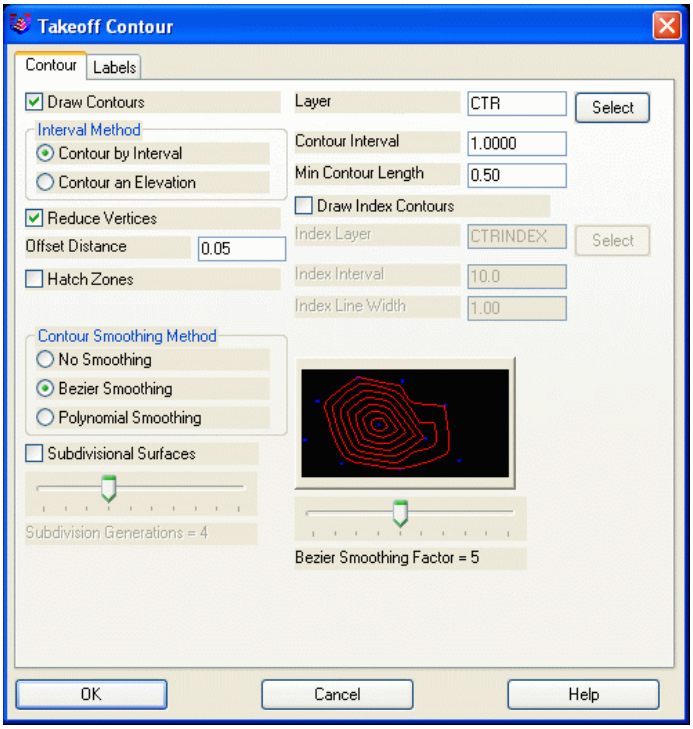
This command allows you to change the features of the different display commands. Note: You can toggle on/off the Existing, Design, and Other surfaces by right clicking with your mouse. To activate this feature type in "shortcutmenu" in the command line and then <1>. To turn off the feature type in <0>.



Display Setup: Here is the master list for the major things you can display, including: the Entities, Contours, and Surface for both the Existing and Design, Cut/Fill Displays, and Other Drawing Entities.



Contour Options: Here you can set the interval, the elevation difference between each contour, for both the Existing, Design by clicking on Design and Existing Contour Settings. The Smooth Contours option will apply smoothing to the contour polylines. You can also choose to draw only the daylight line between Existing and Design instead of the Cut/Fill contours at an interval.



Draw Contours

When this box is checked, the program will draw contour lines after triangulating. Otherwise, only the designated triangulation operations are performed. Specify the layer for contours in the edit box to the right.

Contour by Interval or **Contour an Elevation**
Select whether to contour by interval (ie: every 10 feet) or to contour a certain elevation. The elevation option allows you to contour specific values. For example, if you want just the 100ft contour, then select elevation and enter 100. The default mode is by interval.

Contour Interval

Specify the interval to contour. Note: If the above option is set to Contour an Elevation, then this field is used to specify the elevation to contour.

Minimum Contour Length

Contour lines whose total length is less than this value will not be drawn.

Reduce Vertices

This option attempts to remove extra vertices from the contour polylines which has the advantages of a faster drawing and smaller drawing size. Default is ON

Offset Distance

When the Reduce Verticies option is enabled, This value is the maximum tolerance for shifting the original contour line in order to reduce vertices. The reduced contour polyline will shift no more than this value, at any point, away from the original contour line. A lower value will decrease the number of vertices removed and keep the contour line closer to the original. A higher value will remove more vertices and allows the contour line to shift more from the original.

Hatch Zones

When activated, this option will allow you to hatch the area between the contours sequentially. A secondary dialog will load allowing the user to specify the hatch type and color.

Draw Index Contours

This option creates highlighted contours at a specified interval. When enabled, the fields for Index Layer, Index Interval and Index Line Width are activated.

Contour Smoothing Method

Select the type of contour smoothing to be performed. Bezier smoothing holds all the contour points calculated from the triangulation and only smooths between the calculated points. Polynomial smoothing applies a fifth degree polynomial for smooth transition between the triangulation faces. The smoothing factor described below affects the smoothing bulge.

Bezier Smoothing Factor

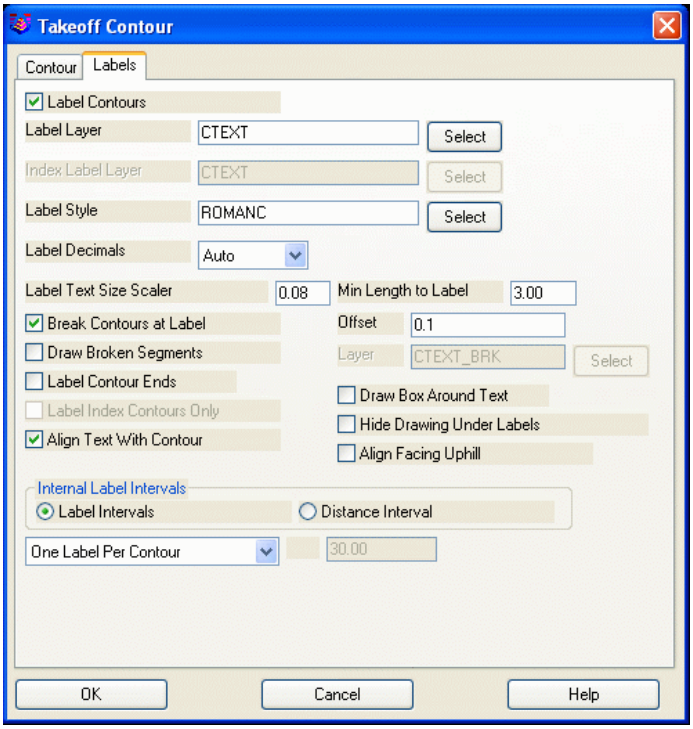
The contour preview window shows you an example of how much smoothing can be expected at each setting. Sliding the bar to the left results in a lower setting which have less looping or less freedom to curve between contour line points. Likewise, moving the slider to the right results in a setting that increases the looping effect.

Subdivisional Surfaces / Subdivisions Generation

This option causes each triangle in the triangulation surface model to be subdivided into an average of three smaller triangles per subdivision generation, with the new temporary vertices raised or lowered to provide smoother contours. More generations increases the smoothness of the algorithm at a cost of increased processing time. If Straight

Lines are chosen as the contouring drawing method, then the contours are guaranteed never to cross. The original points of the surface model are always preserved. These modifications to the surface model are only for contouring purposes and are not written to the triangulation (.FLT) file or inserted into the drawing. If some contour movement is too small for appearance's sake, consider enabling Reduce Vertices.

Label Tab



Label Contours

When activated, contours will be labeled based on the settings below.

Label Layer

Specifies layer name for intermediate contour labels.

Index Label Layer

Specifies layer name for index contour labels.

Label Style

Specifies the text style that will be used for the contour label text.

Label Text Size Scaler

Specifies the size of the contour labels based on a multiplier of the horizontal scale.

Min Length to Label

Contours whose length is less than this value will not be labeled.

Break Contours at Label

When checked, contour lines will be broken and trimmed at the label location for label visibility. When enabled, the Offset box to the right acitvates. The Offset determines the gap between the end of the trimmed contour line and the beginning or ending of the text.

Draw Broken Segments

When checked, segments of contours that are broken out for label visibility will be redrawn as independent segments. Specify the layer for these broken segments in the box to the right of this toggle.

Label Contour Ends

When checked, contour ends will be labeled.

Draw Box Around Text

When checked, a rectangle will be drawn around contour elevation labels.

Label Index Contours Only

When checked, only the index contours will be labeled. This option is active only when "Draw Index Contours" has been selected in the Contour tab of the main dialog.

Hide Drawing Under Labels

This option activates a text wipeout feature that will create the appearance of trimmed segments at the contour label, even though the contour is fully intact. This feature provides the user with the best of both worlds; you have clean looking contour labels, and the contour lines themselves remain contiguous. This feature will also hide other entities that area in the immediate vicinity of the contour label.

Align Text with Contour

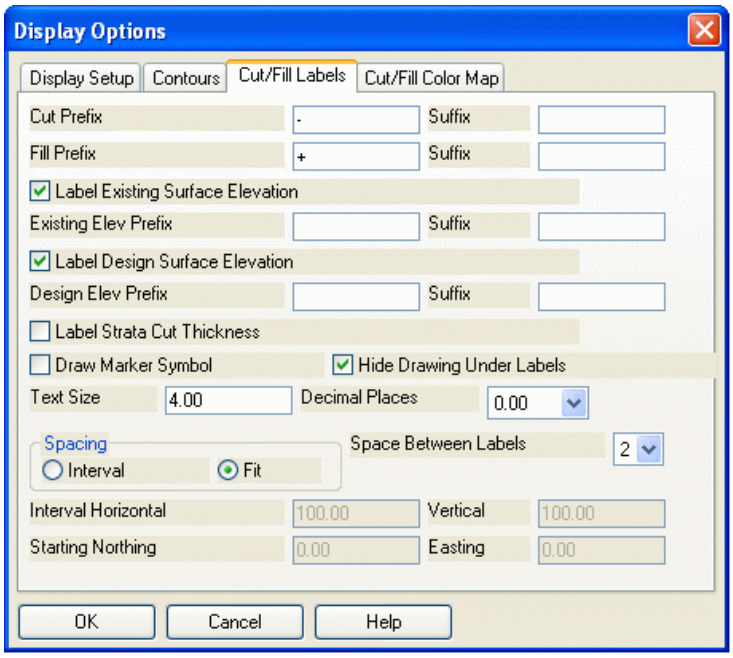
When checked, contour elevation labels will be rotated to align with thier respective contour lines. This option also acitvates the Align Facing Uphill feature explained below.

Align Facing Uphill

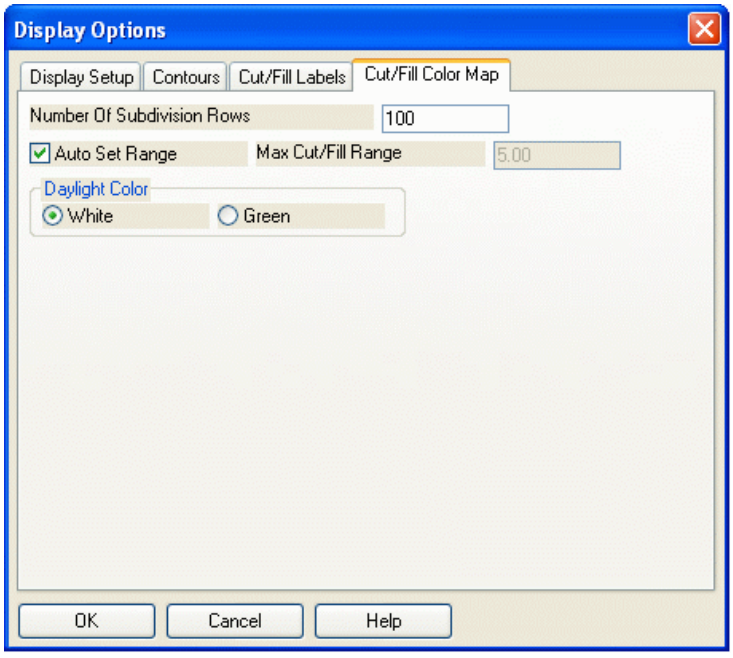
When checked, contour elevation labels will still be rotated to align with thier respective contour lines, but the labels will be flipped in such a manner that the bottom of the text label will always be toward the downhill side of the contours. So as the labels are read right side up, you are always facing uphill.

Internal Label Intervals

Choose between label intervals or distance interval. Label intervals will label each contour with a set number of labels. Distance interval lets you specify a distance between labels.



Cut/Fill Label Options: Here you can customize the Cut/Fill labels. Text can be added either before or after the Cut/Fill amount, the Existing elevation, and the Design elevation with the Prefix and Suffix fields. You can also choose whether or not to display the Existing Surface elevations, the Design Surface elevations and Strata Cut Thickness. Carlson TakeOff gives you the option to draw a marker symbol for where each label represents. You can also hide the drawing under the labels so that you can read the labels clearly. Text Size chooses the text size for each line of the label. Decimal Places sets to how many decimal places the labels will report. The Spacing of the labels can be determined by intervals or by a selected number of spaces. The size of each space is determined by the Text Size.



Cut/Fill Color Map Options: Number Of Subdivision Rows is the number of blocks both horizontally and vertically in the Color Map. If the box reads 100 that means 100 blocks left to right and 100 blocks up and down or 10,000 total pixels. A higher the Number Of Subdivision Rows will make the Color Map sharper, however too high number can cause Carlson TakeOff to run slower. Auto Set Range will automatically set the red to blue scale for your cut/fill levels. However, if you desire greater contrast, then use Max Cut/Fill Range to manual set the range. Use lower numbers for greater contrast. The Daylight Color can be set to be either White or Green.

Prerequisite: a drawing

Keyboard Command: tk_display_options

Window Menu 16

Cascade

Arranges windows so they overlap

Tile Horizontally

Arranges windows side by side as non-overlapping tiles

Tile Vertically

Arranges windows on top of one another as non-overlapping tiles

Arrange Icons

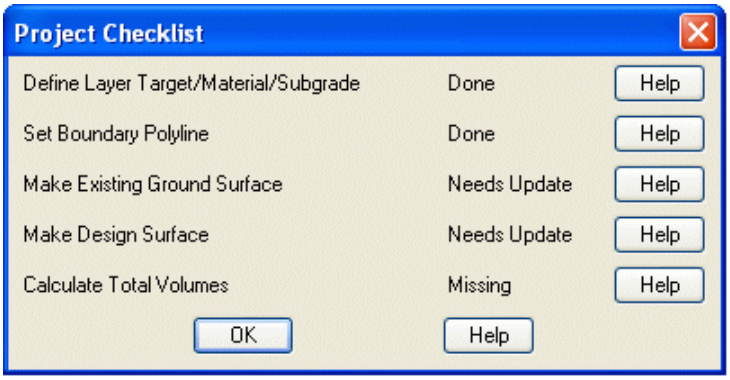
Arranges icons at the bottom of the window

Help Menu 17

Project Checklist

Function

This command allows to check the status of steps needed to calculate total volumes.



Prerequisite: none
Keyboard Command: tk_checklist

OnLine Help

Function

This command opens the Carlson TakeOff on-line Help File.

Prerequisite: None

Keyboard Command: [F1] or HELP

Training Movies

Function

This command opens an application that lets you choose from several training movies. The movies provide instruction for all aspects of Carlson TakeOff.

Prerequisite: None

Keyboard Command: RUN_MOVIES

Carlson WebSite

Function

This command brings you to the Carlson Home webpage.

Prerequisite: an internet connection
Keyboard Command: ._browser Enter Web location (URL) <http://www.carlsonsw.com>:
"http://www.carlsonsw.com/"

About Carlson TakeOff

Function

Displays the Carlson TakeOff version number, serial number, license information, and copyright information. You can run the registration wizard by clicking the Change Registration button on this dialog.

Prerequisite: None

Keyboard Command: ABOUT_SCAD

Index

.GRD file, 494
.PRO, 456, 457, 467–471, 473, 481, 485, 520, 544, 550
2 Tangents Arc Length, 243
2 Tangents Chord Length, 243
2 Tangents Radius, 242
2D Polyline, 230, 285, 317, 444, 446
2D to 3D Polyline by Points, 331
2D to 3D Polyline by Start/End Elevations, 335
2D to 3D Polyline-By Text, 332
2D to 3D-By Text With Leader, 333
3 Point Curve, 243
3D Drive Simutation, 214
3D Entity to 2D, 194
3D Polyline, 232, 248, 250, 284, 318, 444, 446, 449, 469, 498
3D Viewer Window, 221

About Carlson TakeOff, 579
Add Intersection Points, 198
Add Polyline Vertex, 198
Adjust Design Surface, 137
Adjust Overexcavate Surface, 138
Align, 150, 182, 184, 238, 291, 295, 443, 447, 491, 492, 498, 571
Alphanumeric, 290, 382
Angle Mode, 378, 385
Arc, 231–233, 242, 244, 287, 389
Area by Closed Polylines, 374
Area by Interior Point, 374
Area by Inverse, 372
Area by Lines & Arcs, 373, 374
Area Defaults, 371, 373
Areas, 320, 325, 326, 357, 358, 366
Areas Of Interest, 128, 170
ASE, 159–161, 178
Assign Contour Elevations - From Contour Labels, 344
Assign Contour Elevations - Multiple in Series, 342
Assign Contour Elevations - Single Elevation Group, 345
Assign Template Point Centerline, 545, 552
Assign Template Point Profile, 544, 546, 552
Attribute Layout ID, 284, 385
Attributes, 172
Audit, 81
Auto Annotate, 198
AutoCAD Overview, 67
Azimuth, 270, 378, 381, 385

Backfill, 429
Bar Scale, 260
Bearing, 106, 185, 188, 244, 353, 378, 385
Bearing and Distance, 358, 476
Berm Grades, 530
Blips, 197
Block Explode, 182
Boundaries, 241
Boundary, 128, 133, 241, 374
Boundary Polyline, 127, 128, 134–136, 430
Break, 201, 493
Break 3D Polyline by Surface, 202
Break at Intersection, 186
Break At Selected Point, 186
Break by Closed Polyline, 185, 201
Breakline, 284
Breaklines, 145
Building Envelope Polyline, 249

CAD File TakeOff, 2
Calculate End Area, 520, 521, 523
Calculate Offsets, 451, 452
Calculate Pond/Pit Volume, 161
Calculate Sections Volume, 519
Calculate Stockpile Volume, 159
Calculate Total Volumes, 162, 168, 408
Calculate Volumes Inside Perimeter, 168
Carlson_WebSite, 578
Centerline, 20, 329, 434, 440, 444, 447–449, 466, 470, 545, 551, 552
Centerline File, 545, 556
Centroids, 520, 554
Change Elevations, 328
Change Layer, 226, 328
Change Polyline Width, 203
Change Properties, 187
Check Elevation Range, 204
Circle, 233, 511
CLAYER, 225
Clear Overexcavate Surface, 140

Clear Strata Surface, 408
Clipboard, 116, 117, 180
Close, 75, 103, 158, 159, 185, 205, 230–233, 250, 281, 317–319, 321, 322, 445
Closed Polyline By Interior Point, 247
Close Polylines, 205
Closure Error, 360
Color Contours by Elevation, 263
Color Contours by Interval, 264
Compound Curve, 540
Compress CRD File, 291
Configure, 380
Contour Elevation Label, 261
Contour Polyline, 321
Contours, 150, 153, 203, 204, 264, 265, 321, 322, 493, 570
Convert, 101, 105, 121, 189, 192, 237, 301, 305, 339, 342
Convert LDD-AEC Contours, 121
Convert LDD Contours, 108
Convert Points to AECC POINTs, 304
Convert Points to C&G, 305
Convert Points to Eagle Point, 306
Convert Points to Softdesk, 305
Convert Spot Elev To Points, 339
Convert Wild/Leica to Points, 305
Coordinate File, 92, 94, 303
Coordinate File Utilities, 272, 290, 306
Coordinate Transformation, 291, 292
Copy To Layer, 180
Create Points On Surface, 299
Create Trench Network Structure, 415
Crosshairs, 393, 394
Crossing Breaklines, 147
Curve Info, 350
Custom, 79, 80, 240, 401
Cut/Fill, 14, 63, 142, 166, 168, 215, 267, 499, 500, 520, 522, 564, 568, 573
Cut Fill Centroids, 267
Cut Fill Color Map, 565
Cut Fill Contours, 564
Cut Fill Labels, 564
Cut Fill Map Legend, 267

Database, 76, 382
Data Collectors, 85, 87
Data Files in DWG Directory, 381
Decimal Places, 470, 481, 517, 572
Define Layer Target/Material/Subgrade, 123, 126, 134, 142, 168, 171, 314, 355, 560, 562, 566, 567
Define Materials, 125, 170
Definition, 42, 237, 277, 280, 282, 398
Densify Polyline Vertices, 196, 197
Density, 125, 126, 131, 399
density, 42, 121, 157, 160, 399
Depth Contours, 409, 410
Design, 5, 8, 59, 123, 137, 138, 143, 168, 179, 214, 253, 270, 314, 321, 337, 351, 352, 360, 383, 389, 415, 417, 418, 423, 428, 430, 457, 479, 486, 506, 524, 526, 528, 529, 533, 536, 541, 542, 544, 545, 550, 551, 553, 554, 556, 562, 568, 572
Design Contours, 562, 563
Design Drawing, 562
Design Pad Template, 250, 251, 253, 255–257
Design Road Profile, 454, 550
Design Sewer/Pipe Profile, 456
Design Surface, 563, 564, 572
Design Surface 3D Viewer, 217
Design Surface Vertical Offset, 142
Design Template, 524, 528, 541, 543–545, 549–551
Digitize Menu, 307
Digitizer Settings, 312, 313
Digitizer_Setup, 308
Digitizing, 47, 50, 52, 308, 322
Display Last Report, 117
Display Menu, 559
Display Options, 567
Display Order, 213
Distance Report, 358
Ditch, 529, 530, 538
Ditch Grades, 529, 530
Dozer, 109, 189
Drape 3D Polyline On Surface, 338
Draw, 3–6, 10, 13, 27, 41, 45, 48, 60, 61, 63, 74–76, 79, 81, 108, 115, 118, 136, 148, 150, 156, 173, 174, 184, 236–238, 242, 250, 262, 266, 267, 272, 286, 290, 293–295, 320, 326, 351–353, 361, 366, 378, 379, 383, 384, 388, 389, 393, 395, 429, 434, 436, 440, 465, 468, 475–479, 481, 487, 488, 498, 508, 515, 534, 553, 555, 569, 571
Draw-Locate Points, 270, 273, 290
Draw 3DPoly Base Breakline, 159–161
Draw 3DPoly Perimeter, 158
Draw Centerline File, 440
Drawing Cleanup, 120, 122
Drawing Inspector, 352, 353, 358

Drawing Setup, 162, 258, 260, 273, 284, 378, 380, 381, 383, 443, 447, 506, 515, 516
Draw Menu, 229
Draw Overexcavate Cut Color Map, 139
Draw Overexcavate Surface 3D Faces, 139
Draw Polyline Blips, 197
Draw Profile, 473, 480, 522
Draw Section File, 506, 549, 550, 553
Draw Strata Cut Color Map, 410
Draw Strata Cut Depth Contours, 409
Draw Strata Surface, 411, 420
Draw Trench Network Plan View, 419
Draw Trench Network Profile, 419
Draw Triangular Surface, 265
Draw Typical Template, 534
Draw Typical Trench Template, 427
Drillhole and Strata, 41, 398
Drillhole Import, 400
Drillhole Menu, 397
Drillhole Strata Settings, 398
Duplicate Strata, 402

Eagle Point, 69, 281, 306
Edit, 9, 34, 43, 44, 109, 124, 126, 170–173, 200, 213, 233, 258, 278, 282, 285, 346, 395, 396, 398, 399, 403, 405, 425, 434, 436, 476, 491, 502, 503, 536, 539, 542
Edit-Assign Polyline to Elevations, 329
Edit Contours, 345
Edit Drillhole, 404
Edit Menu, 177
Edit Point, 295
Edit Polyline Section, 199
Edit Polyline Vertex, 198, 330
Edit Symbol Library, 385
Edit Text, 188
Edit Trench Network Structure, 417
Edit_Selected_Layer, 126
Elevate Menu, 327
Elevation, 3, 10, 31, 33, 51, 52, 54, 57, 106, 109, 122, 145, 148, 151, 155, 158–161, 215, 216, 218, 271, 272, 290, 293, 294, 296, 298, 315, 316, 318, 319, 321–325, 328–330, 336, 343, 353, 361, 366, 367, 383, 384, 399, 403–405, 414, 416, 423, 444, 452, 455, 465, 479, 488, 496, 504, 513, 530, 552, 569
Elevation Difference, 385
Elevation Zone Analysis, 366, 368
End Areas, 325, 488, 511, 521, 523
Entities to Polylines, 195

Erase, 9, 121, 128–130, 145, 180, 202, 207, 290, 293, 379, 469, 480
Erase, Select, 178
Erase by Closed Polyline, 179
Erase by Layer, 178, 179
Erase Outside, 180
Erase Overexcavate Cut Color Map, 140
Erase Overexcavate Surface 3D Faces, 139
Erase Points, 289
Erase Strata Cut Color Map, 410
Erase Strata Cut Depth Contours, 409
Erase Strata Surface, 411
Erase Subgrade Hatches, 357
Erase Subgrade Labels, 357
Existing, 314, 321, 420, 501, 502, 538, 549, 551, 553, 556, 557, 567
Existing Contours, 560, 561
Existing Drawing, 560
Existing Surface, 561, 562
Existing Surface 3D Viewer, 216
Existing Surface Vertical Offset, 143
Exit, 118, 323, 324, 354, 387, 426, 435, 473, 495, 542
Export Surface DXF Files, 110
Export Text/ASCII File, 303
Export Topcon Tin File, 111
Extend, 184, 230–232, 392, 493, 521
Extend by Distance, 184, 185
Extend To Edge, 184
Extract Project Archive, 82

Factor, 131, 253, 388, 520, 553
Field to Finish, 273, 274, 278, 279, 285, 287
File Menu, 73
Fillet, 192
filters, 112, 120
Find Trench Network Structure, 419
FlyOver Along 3D Polyline, 219
Freeze Layer, 227

General Commands, 70
Geodimeter, 86, 104–106
GPS, 276
Grid, 39, 91, 251, 252, 322, 323, 420, 457, 467, 474, 475, 477, 479, 480, 488, 494, 508, 512
Grid File Utilities, 109
Guardrail Expansion, 530

Hatch, 239–241, 355, 357, 362, 366, 428, 569
Hatch Subgrade Areas, 355
Header, 290, 472
Help Menu, 577

Highway Section Staging, 500

Image Adjust, 191
Image Clip, 190
Image Frame, 189
Import-Export, 85
Import/Export Carlson Triangulation Files, 111
Import/Export DXF Files, 108
Import/Export LandXML Files, 114
Import Centerline, 441
Import LDD MDB File, 303
Import Profile, 468
Import Sections, 497
Import TDS RD5 File, 302
Import Text/ASCII File, 301
Import Xref to Current Drawing, 83
Info, 105
Initialize Overexcavation Surface, 137
Input-Edit, 174, 434, 435, 492, 493, 548, 553
Input-Edit Centerline File, 434
Input-Edit Profile File, 471
Input-Edit Section Alignment, 489, 491
Input-Edit Section File, 502, 506, 519
Input-Edit Super Elevation, 539, 552
Input-Edit Template Series, 541
Input Edit End Area File, 521
Input Edit Trench Template, 425
Input Trench From Polyline, 414
Inquiry, 6, 9, 10, 55, 63, 134, 136, 354, 385, 393
Inquiry Menu, 347
Insert Drawing, 235
Insert Symbols, 235
Interpolate Points, 297, 298
Intersections, 135
Inverse with Area, 371
Isolate Layer, 227, 270
Issuing Commands, 68

Join, 13, 156, 189, 192, 269, 270, 277, 344
Join Nearest, 192, 195

Label, 6, 45, 128, 130, 132, 142, 149–151, 262, 263, 271, 287, 371, 373–375, 410, 419, 442–445, 447, 448, 477–479, 481, 482, 509, 510, 560, 562, 570, 571
Label Contours, 149, 570
Label Station-Offset, 447, 448
Label Subgrade Areas, 357
Label Zeros, 271, 287
Land Development Desktop, 121, 304
Layer, 4–6, 8, 9, 16, 30, 50, 58, 59, 61, 69, 71, 72, 112, 123, 134, 136, 138, 141, 149, 150, 156, 180, 187, 224–226, 253, 262, 266, 270–272, 283, 285, 287, 348, 349, 352, 353, 371, 415, 418, 423, 441, 447, 456, 493, 569, 570
Layer Control, 224
Layer ID, 349
Layer Report, 354
Layers, 71, 179, 224, 225, 247, 284, 480, 515
Layer_Inspector, 354
LDD, 121, 304
LDD MDB, 303
least squares, 291, 292
Leica, 86, 99–103, 306
Length, 109, 148, 150, 243, 244, 389, 438, 442, 446, 569, 570
Line, 3, 18, 68, 76, 102, 112, 121, 146, 149, 179, 186, 187, 195, 225, 230, 232, 242, 243, 248, 265, 266, 281, 283, 285, 287, 288, 290, 292, 302, 341, 350, 371, 373, 378, 382, 384, 394, 402, 424, 434–438, 442, 446, 448, 450, 464, 466, 470, 476, 478, 479, 481, 491, 492, 494, 506, 551, 569, 570
Linear, 124
LISCAD, 101, 102
LisCad, 306
List, 348, 394, 401, 418, 495, 502, 503, 505
List Elevation, 10, 358
Load Tablet Calibration, 312
Locate on Real Z Axis, 271, 384
Locate Point, 270, 272

Make, 45, 88, 98, 123, 136, 160, 162, 253, 362, 366, 368, 403, 405, 489, 494, 522, 546
Make Design Surface, 136, 142
Make Existing Ground Surface, 134, 135, 142
Make Overexcavate Surface, 138
Make Strata Surface, 408–411
Make User-Defined Surface, 141, 142
Manhole, 32, 414, 415, 421, 456–458, 460, 464
Mass Diagram Output File, 520, 555
Material Quantities, 168
Measured, 102
Merge Existing With Design, 143
MicroStation, 109
Mouse Clicking Settings, 393
Move, 5, 6, 30, 42, 68, 70, 123, 180, 184, 185, 215, 217, 218, 230, 269, 290, 293, 357, 400, 401, 404, 434, 524

NADCON, 291
Nearest Found, 274
New, 74, 85, 102, 155, 188, 190, 194, 198, 204, 224, 241, 277, 284, 288, 387, 457, 467, 493, 521, 538, 550
Nikon, 86, 103, 104
Nodes, 271
North Arrow, 260
Numeric, 382

Object Linking, 381
Object Snap, 390, 391, 393
Offset, 34, 106, 137, 148, 150, 155, 201, 324, 329, 330, 383, 417, 445, 448, 449, 463, 464, 482, 485–488, 490, 494, 498, 502, 503, 508, 509, 511, 514, 515, 530, 532, 533, 546, 569, 571
Offset Cutoff, 3
Offset Point Entry, 449
Offset Polyline, 193
OLE objects, 76
On-Screen Edit Points, 289
OnLine Help, 578
Open, 75, 80, 100, 109, 115, 185, 205, 221, 263, 305, 308, 326, 465, 502
Open Polylines, 205
Options, 14, 27, 63, 71, 74, 79, 89, 96, 116, 151, 158, 159, 231–233, 235, 261, 263, 308, 359, 362, 444, 447, 459, 486, 487, 490, 542, 560, 562, 564, 565, 568, 572, 573
Origin, 117, 153
Other, 314, 328, 362, 367, 371, 379, 383, 385, 390, 450, 454, 480, 490, 560, 562, 566, 567, 569
Other Drawing, 566, 567
Output Coordinate File, 553
Overexcavate Surface, 137, 138, 486

Pad Polyline By Interior Text, 336
Pan, 212, 222, 308, 329, 381, 434, 465, 472, 503
Parameters, 26, 513, 514, 555, 556
PC PT Radius Point, 244
PC Radius Chord, 244
PDMODE, 381
PDSIZE, 381
Perimeter, 319, 320, 374, 383, 490
Perimeter Polylines Properties, 194
Perimeter_Polylines_Properties, 193
Pipe, 34, 39, 422, 424, 456, 460, 464, 484
Pipe Depth Summary, 483
Pipe Size, 32, 34, 416
Pit, 145
Place Drillhole, 42, 43, 45, 400, 402, 408
Plain View Label Settings, 423
Plot, 76–80, 225, 226, 292, 361, 366, 367, 378, 384, 440, 441, 488, 495, 508, 511, 521, 555
Point, 314–316, 343, 350–353, 359, 368, 381, 382, 384, 385, 395, 435, 436, 442, 444, 446, 449, 450, 458, 470, 471, 478, 480, 482, 485, 487, 490, 498, 519, 521, 526, 529–531, 536, 541, 544–546, 550, 552
Point Defaults, 298
Point Description, 297
Point ID, 349
Point Layer, 285
Point Number Format, 290
Point Protect, 92, 95–97, 99, 100, 103, 104, 116, 296, 302
Point Range, 91
Points, 49, 233, 241, 242, 268, 278, 281, 282, 284, 286–288, 291, 293–299, 305, 310
Polyline, 3, 10, 11, 13, 17, 18, 20, 30–32, 50, 52, 53, 55, 58, 109, 159, 161, 179, 181, 186, 194, 196, 198, 231, 232, 253, 271, 317–320, 328, 329, 350, 353, 359, 360, 373, 374, 414, 415, 440, 445, 448–450, 453, 454, 465, 466, 482, 487, 489, 495, 553, 555
Polyline File, 110
Polyline Info, 350
Polyline Report, 359
Polylines, 8, 10, 109, 110, 121, 127, 128, 133, 151, 154, 186, 192, 194, 198, 202, 203, 262, 330, 428, 490, 555
Polyline Slope Report, 481
Polyline to Centerline File, 440
Pond, 161
Precision, 284, 371, 389
Preferences, 379, 389
Prepare, 523
Print Earthwork File Report, 523
Process, 13, 27, 86, 96, 107, 112, 122, 206, 281, 283, 286, 302, 333, 335, 341, 342, 344, 361, 362, 470, 493, 494, 496, 498, 520, 524, 526, 541, 556
Process Road Design, 549, 550, 555
Profile, 18, 19, 31, 39, 114, 250, 253, 414, 420, 444, 449, 450, 453–457, 465–467, 469–473, 475–477, 479–481, 483, 485, 544, 548, 556
Profile from 2D Polyline, 467
Profile from Design Surface, 454
Profile From Existing Surface, 453
Profile from Grid or TIN File, 467

Profile from Points on Centerline, 468
Profile from Screen Entities, 466
Profile Report, 485
Profile To 3D Polyline, 469
Profile To Points, 469
Project Checklist, 578
Properties, 72, 187, 224, 225, 242, 278
Properties and Layers, 71
Purge, 81

Quantities, 428, 526
Quick Profile, 45, 142
Quick Profile from Screen Entities, 464, 465
Quick Section, 19, 486

Railroad, 490
Raster Image, 245
Recover, 80
Redo, 178
Redraw, 210, 491
Reduce Polyline Vertices, 109, 121, 196
Reduce Vertices, 148, 149, 569, 570
Regenerate, 210
Registration, 579
Remove, 3, 70, 81, 115, 120, 121, 140, 172, 181, 190, 242, 291, 302, 381, 400, 401, 404, 405, 434, 472, 524, 542
Remove Duplicate Polylines, 200
Remove Groups, 191
Remove Polyline Arcs, 201
Remove Polyline Segment, 201
Remove Polyline Vertex, 201
Remove Trench Network Structure, 418
Removing, 201
Rendering, 222
Renumber Points, 293
repackaged, xiv
Report, 9, 16, 61, 62, 65, 115, 116, 122, 124, 129, 136, 162, 163, 170, 171, 254, 269, 282, 294, 295, 368, 381, 406, 407, 429, 452, 484, 485, 517–520, 522, 554
Report Log, 380
Reports, 406
Report Trench Network, 430
Restore, 225, 396
Restore Due North, 213
Restore Layer, 227
Reverse, 185, 196
Reverse Polyline, 196
Right of Way, 530
Road Design, 25–27, 250, 253
Roads Menu, 433
Road Work, 16
Rotate by Bearing, 187
Rotate Points, 295
Runoff, 266
Run Off Tracking, 266
RW, 86, 90, 93, 94, 96, 103–105, 107, 274

Save, 5, 8, 9, 19, 25, 30, 34, 38, 44, 48, 59, 63, 75, 82, 84, 85, 141, 156, 170, 225, 246, 277, 284, 285, 323, 324, 381, 396, 402–404, 406, 419, 435, 439, 465, 473, 491, 506, 515, 539, 542
Save As, 75
Save Tablet Calibration, 312
Scale, 28, 38, 49, 79, 80, 85, 182, 183, 187, 215, 216, 218, 222, 236, 240, 247, 258, 262, 295, 310, 311, 378, 381, 384, 473, 475, 476, 478, 479, 482, 488, 504, 508, 511, 521
Screen Pick Point, 272
Section File, 323, 487, 493, 494, 498, 513, 519, 550, 551, 553
Section Name, 506
Section Report, 517, 519, 555
Sections, 322, 488, 489, 493, 494, 501, 505, 506, 508, 512, 514, 520, 551, 555
Sections From Design Surface, 491
Sections From Existing Surface, 491
Sections from Grid or TIN File, 494
Sections from Points, 496, 551
Sections from Polylines, 494
Sections from Screen Entities, 492
Sections to 3D Polylines, 497
Sections to Points, 498
Select by Area, 207
Select by Filter, 205
Selection of Items, 69
Select_by_Elevation, 206
Set Active Surfaces, 142
Set Environment Variables, 394
Set Layer, 17, 227
Set Layer For Design, 127
Set Layer For Existing, 126
Set Layer For Other, 127
Set Point Elevations To Surface, 300
Set Polyline Origin, 197
Set Polyline to Elevation, 328
Settings Menu, 377
Set UCS to World, 394
Sewer, 31, 414, 420, 456–458, 472

Sewer Annotation, 459
Sewer Network, 420
Sewer Structure, 34, 415–417, 426
Shade, 139, 222
ShrinkWrap Entities, 248
Sight Distance, 456, 473, 480
Sketch, 51, 321, 322
Slope, 19, 21, 24, 34, 54, 55, 158, 159, 215, 250, 251, 253, 319, 330, 358, 361–363, 366, 367, 424, 430, 445, 456, 457, 465, 472, 481, 485, 488, 493, 500, 510, 519, 528–532, 544, 553, 555
Slope Area, 361
Slope Distance, 105, 482
Slope Ratio, 482
Slope Report, 360, 362, 481, 482
Slope Zone Analysis, 361, 363
Slope Zone Section Analysis, 499
SMI, 86, 98, 99
SMI Chain File, 554
Smoothing, 109, 149, 569
Smooth Polyline, 203, 284
Snap Tolerance, 336
Softdesk, 305, 383
Sokkia/SDR, 95, 96
Sokkia G, 86, 97
Special, 90, 274, 276, 282, 452, 533
Spiral, 436–438
Spline, 121
Spot Elevation, 315, 316
Spreadsheet Edit Points, 288
SRVPNO, 284, 348, 385
Standard, 16, 65, 87, 106, 109, 168, 169, 171, 324, 325, 406
Standard Copy, 180
Standard Explode, 181
Standard Rotate, 188
Startup Wizard, 2, 16, 29, 47, 380
Station-Offset, 449, 555
Station/Elevation, 485
Station Polyline/Centerline, 441
Step, 2–4, 8, 10, 13, 14, 16–19, 25, 28, 29, 41, 42, 45, 47, 55, 57, 58, 61, 63, 65
Stockpile, 158–161
Store Project Archive, 82
Story Stake By Points/Polyline, 370
Story Stake From Surface Entities, 368
Subgrades, 125, 428
Surface, 13, 14, 17, 30, 45, 52, 55, 63, 123–125, 131, 136, 137, 140–142, 144, 145, 149, 153–155, 157–159, 202, 215, 219, 221, 351, 352, 362, 366, 367, 404, 405, 408, 409, 414, 429, 473, 483, 493, 526, 569
Surface Area, 363, 367
Surface Entities, 196, 466, 479, 481
Surface Inspector, 45, 142, 350
Surface Manager, 140
Surface Model, 221, 250, 351, 466, 505
Surface Report, 351
Surface_Report, 351
SurvCE, 86, 87, 90, 92, 94, 97
Surveyor, 94, 95, 389
Surveyors Assistant, 86
SurvStar, 86, 91
Symbol Library, 233, 260
Symbols, 233
Symbol Size, 136, 284, 381, 384, 398

Tablet Calibrate, 310
Tablet Calibration, 48, 310, 311
Tablet Off, 309, 313, 319
Tablet On, 309, 312
Tag Hard Breakline Polylines, 202
TDS, 86, 97, 98, 301, 302
Technical Support, 313
Template, 18, 20, 34, 47, 74, 76, 196, 250, 277, 284, 425, 427, 428, 483, 534, 536–538, 545, 554, 556
Template Transition, 536, 539, 552
Terramodel, 441, 469
Text, 11, 76, 91, 129, 150, 152, 181, 188, 189, 238, 262, 263, 271, 284, 290, 301, 340, 341, 353, 378, 384, 442, 443, 447, 448, 456, 461, 476, 479–482, 487–489, 495, 497, 511, 512, 514, 520, 521, 554, 570–572
Text EnlargeReduce, 188
Text Explode To Polylines, 109, 189
Text Size, 150, 188
Text Style, 387, 388
Thaw Layer, 227
Title Block, 258, 259
TO, 17, 86, 98, 99, 108, 126, 194, 195, 227, 354, 395, 404, 529, 542
Toolbars, 386
Tools Menu, 119
Topsoil, 131–134, 299, 300, 320, 543, 553
Topsoil Removal, 130, 133, 299, 300, 351, 352, 486, 550, 552, 553
Topsoil Removal/Replacement, 134, 136, 193, 195, 542, 552
Topsoil Removal and Replament Options, 130

TR-, 233
Training Movies, 578
Transit, 528, 530, 537, 538
Translate Points, 295
Trench Menu, 413
Trench Network, 28, 30, 32–35, 37, 38, 45, 419, 426, 430
Trench Network File Backup, 419
Trench Network Quantities, 428
Trench Subgrade Areas, 428
Triangulate & Contour, 143, 147, 151, 157, 202, 203, 265, 271, 284, 492, 551, 555, 556
Triangulation, 28, 110–112, 114, 144, 153, 157, 158, 265, 266, 361, 383, 494
Triangulation File Utilities, 153
Trim, 27, 183, 301, 336, 556
Trimble, 86, 95
TS, 6, 92, 97, 225, 258, 272, 283, 299, 348, 352, 390, 436, 437, 481, 506, 514, 542
Tsunami, 279
Tutorial, 1, 2
Twist, 381, 448
Twist Screen Line, 213
Twist Screen Standard, 212, 213
Twist Screen Surveyor, 213
Two Surface, 253
Typical Template, 524, 525, 536

Undo, 178, 183–185, 190, 200, 230, 293, 317–320, 322–324, 345, 372, 455, 456
Units Control, 388
Universal Transverse Mercator, 291
Untag Hard Breakline Polylines, 203
Update, 214, 254, 290, 293, 306, 383
Update Colors For Set Elevations, 383
Update Drawing from CRD File, 293
Update_Colors_For_Set_Elevations, 214

Vertical Angle, 105, 106, 385
Vertical Angle Mode, 385
Vertical Curve, 19, 456, 473
View, 6, 11, 13, 14, 28, 31–33, 41, 69, 79–81, 84, 139, 144, 145, 162, 163, 215, 222, 223, 225, 242, 245, 254, 359, 379, 381, 383, 455, 457, 465, 476, 487, 503, 523, 540
View Menu, 209
View Overexcavate Surface, 138
Viewpoint 3D, 223, 257, 558
Volume, 15, 35, 45, 46, 64, 127, 142, 144, 154, 157–162, 166, 253, 428, 483, 499, 520, 523, 553
Volumes By Triangulation, 157

Wild, 103, 272, 302
Wildsoft, 101–103, 293
Window, 575
Write Block, 236, 237
Write Polyline File, 109, 189

Xref Manager, 83, 84

Zeiss, 303
Zone Polylines, 109
Zoom - Center, 211
Zoom - Dynamic, 210
Zoom - Extents, 211
Zoom - Previous, 210
Zoom - Window, 210
Zoom IN, 211
Zoom OUT, 211
Zoom Point(s), 212